Tekla Structures 2022

Reference

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1 Advanced options reference

Advanced options customize your version of Tekla Structures. For example, they set various file names and locations, define symbols used in drawings, change how numbering is done, and so on.

Check and change advanced option values

Most advanced options are available in the user interface. To access the options, click **File menu** --> **Settings** and in the **Settings** area select **Advanced options**.

Some advanced options are not shown in the **Advanced Options** dialog box and can only be set in initialization files; the documentation of the option mentions when this is the case.

The default values given for the advanced options in the documentation are for the Default environment. The values may be different in localized environments. If you want to change the defaults used in your projects, add the advanced options in an initialization file that is located in a project or firm folder. Settings in the initialization files do not change existing selections in models. They are used when you create a new model and when you add values for options that have no previous value in the model.

To list all advanced option values set in the current model, including those set in initialization files, click the **Write to file** button at the bottom of the **Advanced options** dialog box.

Find the documentation for advanced options

To see the documentation for those advanced options that are shown in the **Advanced options** dialog box, select an advanced option in the dialog box and press F1 on your keyboard. This opens the correct help page for the option, and often provides additional information even for those options that have a short description directly in the dialog box.

Here in the documentation, we list the advanced options alphabetically (see the attached listing), ignoring the initial letters XS. For example, under **A** you will find the advanced option `XS_AISC_WELD_MARK`, under **B**, `XS_BACKGROUND_COLOR`, and so on. Advanced options starting with `XSR` are listed under **R**.

TIP For instructions on how to find advanced options documentation in Tekla User Assistance, see How does the search work.

1.1 Advanced options - A

XS_AD_ANALYSIS_PLANES_ENABLED

Category: Analysis & Design

Set this advanced option to `FALSE` to revert to the analysis model creation method used in Tekla Structures 2017i and earlier versions.

The default value is `TRUE`.

If you change the value, Tekla Structures recreates the analysis models.

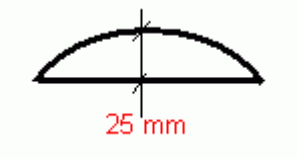
This advanced option is model specific and the setting is saved in the options database.

XS_AD_CURVED_BEAM_SPLIT_ACCURACY_MM

Category

Analysis & Design

Use to set the maximum distance between a curved member and straight segment. Enter the value in millimeters. The default value is `25.0` mm.

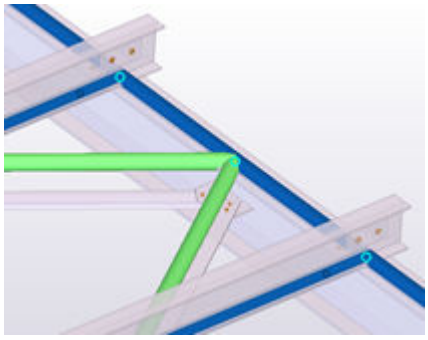


This advanced option is model specific and the setting is saved in the options database.

XS_AD_DRAW_BAR_DIAMETER_MM

Category: Analysis & Design

Use this advanced option to define the diameter of the analysis parts when the analysis parts are shown in model views. Enter the value in millimeters. The default value is 70 mm.



This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Redraw the model views after changing the value to activate the new setting.

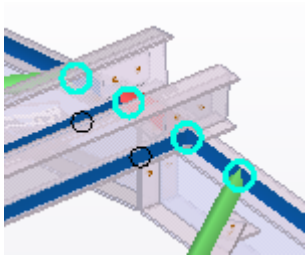
See also

XS_AD_DRAW_NODE_SCALE (page 61)

XS_AD_DRAW_NODE_SCALE

Category: Analysis & Design

Use this advanced option to scale the size of the analysis nodes when the analysis nodes are shown in model views. The default value is 1.



This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Redraw the model views after changing the value to activate the new setting.

See also

XS_AD_DRAW_BAR_DIAMETER_MM (page 60)

XS_AD_ELEMENT_ANGLE_CHECK_ANGLE_DIFF_LIMIT

Category

Analysis & Design

Use to set a limit angle between physical part and analysis model member, in order to test the validity of the analysis model. When the angle is greater than the limit, a message appears and a warning is added to the log. Small differences in angles are usual in the analysis model of a truss, for example. The default value is `10.0`.

This advanced option is model specific and the setting is saved in the options database.

XS_AD_ENVIRONMENT

Category

Analysis & Design

This advanced option is used in analysis and design integration to set the environment, for example, to set the cross section catalog. The default value is `Europe`.

Possible values are dependent on the analysis application that is used. For some applications, this advanced option is not used.

This advanced option is model specific and the setting is saved in the options database.

XS_AD_GET_MOMENT_CONNECTION_STATUS

Category

Analysis & Design

Use to specify to which value (**Yes** or **No**) the **Moment connection symbols** option is set when the **Get results** command is used in the analysis model. The **Moment connection symbols** option is available on the **End conditions** tab in the user-defined attributes properties dialog box of the part, and it defines whether the moment connection symbols are shown in drawings.

Set the advanced option to one of the following values:

- `yz`: if the rotations ry and rz are fixed in the analysis model, the value **Yes** is used, otherwise **No** is used.
- `xyz`: if all rotations are fixed in the analysis model, the value **Yes** is used, otherwise **No** is used.
- `z`: if the rotation rz is fixed in the analysis model, the value **Yes** is used, otherwise **No** is used.
- The advanced option is not set: The value you have set manually in the **Moment connection symbols** option is used.

The default is `yz`.

This advanced option is model specific and the setting is saved in the options database.

XS_AD_GET_RESULTS_DESIGN_VALUES

Category

This advanced option must be set in an initialization (.ini) file.

Set this advanced option to `TRUE` to import steel and concrete design values from Robot into Tekla Structures when you use the **Get results** or **Get results for selected** command in the **Analysis & Design Models** dialog box. The default value is `TRUE`. If you do not want to import the design values, set this advanced option to `FALSE`.

See also

XS_AD_GET_RESULTS_FORCES (page 63)

XS_AD_GET_RESULTS_FORCES

Category

This advanced option must be set in an initialization (.ini) file.

Set this advanced option to `TRUE` to import forces from Robot into Tekla Structures when you use the **Get results** or **Get results for selected** command in the **Analysis & Design Models** dialog box. The default value is `FALSE`.

See also

XS_AD_GET_RESULTS_DESIGN_VALUES (page 63)

XS_AD_LOAD_COMBINATION_METHOD

Category: Analysis & Design

Use this advanced option to define whether a load combination is created instead of a repeated load in STAAD.Pro export from Tekla Structures.

Set to `0` to use `REPEAT LOAD`, or set to `1` to use `LOAD COMBINATION`.

The default value is `0`.

This advanced option is model specific and the setting is saved in the options database.

XS_AD_MEMBER_NUMBER_VISUALIZATION

Category

Analysis & Design

Use to show or hide member numbers when displaying the analysis model in a model view. The default is `TRUE`, which shows the numbers. To hide the numbers, set to `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_AD_MEMBER_RESULT_DISP_DIVISION_COUNT

Category

Analysis & Design

Use to define the analysis member points whose results are saved in the `analysis_results.db5` database. This advanced option defines into how many parts each analysis member is divided.

Enter an integer value. The default is `0`, which does not save any displacement results.

This advanced option is model specific and the setting is saved in the options database.

Example

To save only the displacements of member ends, set `XS_AD_MEMBER_RESULT_DISP_DIVISION_COUNT` to `1`.

To save three intermediate displacement results, in addition to the end displacements, set `XS_AD_MEMBER_RESULT_DISP_DIVISION_COUNT` to `4`.

See also

XS_AD_MEMBER_RESULT_MIN_DISTANCE (page 65)

XS_AD_MEMBER_RESULT_DIVISION_COUNT (page 65)

XS_AD_MEMBER_RESULT_DIVISION_COUNT

Category

Analysis & Design

Use to define the analysis member points whose results are saved in the `analysis_results.db5` database. This advanced option defines into how many parts each analysis member is divided.

The default is `1`, which saves only the results of member ends.

This advanced option is model specific and the setting is saved in the options database.

Example

To save the results of three intermediate points (quartiles) on each member, in addition to the end results, set `XS_AD_MEMBER_RESULT_DIVISION_COUNT` to `4`.

See also

XS_AD_MEMBER_RESULT_MIN_DISTANCE (page 65)

XS_AD_MEMBER_RESULT_DISP_DIVISION_COUNT (page 64)

XS_AD_MEMBER_RESULT_GRID_SIZE

Category

Analysis & Design

Use to define the grid spacing for the analysis results of plates, slabs, and panels. Enter the value in millimeters. The default value is `500`.

This advanced option is model specific and the setting is saved in the options database.

XS_AD_MEMBER_RESULT_MIN_DISTANCE

Category

Analysis & Design

Use to define the minimum distance between the analysis result points on analysis members. Enter the value in millimeters.The default value is `500`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_AD_MEMBER_RESULT_DIVISION_COUNT (page 65)

XS_AD_MEMBER_RESULT_DISP_DIVISION_COUNT (page 64)

XS_AD_MEMBER_TYPE_VISUALIZATION

Category: Analysis & Design

Use colors to show different analysis part types when displaying the analysis model in a model view. The default setting is `TRUE`, which shows the analysis parts using the analysis class (page 819) colors. A setting of `FALSE` shows the plate objects using the analysis class colors and the other analysis parts in blue.

This advanced option is model specific and the setting is saved in the options database.

XS_AD_NEAR_NODES_WARNING_LIMIT

Category

Analysis & Design

Use this advanced option to trigger a warning when analysis nodes are closer to each other than the limit.

Enter the limit in millimeters. The default value is `0`, which means that Tekla Structures does not show warnings.

This advanced option is model specific and the setting is saved in the options database.

Example

To have Tekla Structures show a warning for analysis nodes that are closer than 5 mm to each other, set `XS_AD_NEAR_NODES_WARNING_LIMIT` to `5`.

See also

XS_AD_SHORT_MEMBER_WARNING_LIMIT (page 69)

XS_AD_SHORT_RIGIDLINK_WARNING_LIMIT (page 69)

XS_AD_NODE_NUMBER_BY_Z

Category: Analysis & Design

If you set this advanced option to `TRUE`, the start number of a new analysis node is set according to the global z coordinate of the node. For example:

- If z is less than 1000.0, the node start number is 0.
- If z is between 1000.0 and 1999.0, the node start number is 1000.
- If z is between 2000.0 and 2999.0, the node start number is 2000.

The first free number above the start number is assigned to the node.

The default is `FALSE`.

XS_AD_NODE_NUMBER_VISUALIZATION

Category

Analysis & Design

Use to show or hide node numbers when displaying the analysis model in a model view. The default is `TRUE`, which shows the numbers.

This advanced option is model specific and the setting is saved in the options database.

XS_AD_OPTIMISATION_DISABLED

Category

Analysis & Design

Use to check profile catalog analysis values. To enable design optimization, use the value `FALSE`. To disable design optimization, use the value `TRUE`. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_PROFILE_ANALYSIS_CHECK_ALL (page 358)

XS_AD_OPTIMISATION_NO_WEIGHT_SORT

Category

Analysis & Design

When optimization cross section group is extracted from the profile catalog, the group is sorted by default according to the section weight (cross section area). Set this advanced option to `TRUE` to disable this sorting, in which case the section order is the same as in the profile catalog.

If you use the value `FALSE`, the group is sorted according to the section weight. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_AD_OPTIMISATION_RECURSE_CATALOG

Category

Analysis & Design

Set this advanced option to `TRUE` to have Tekla Structures ignore profile catalog rules for optimizing steel part sizes and search the entire profile catalog. If you want to take the rules into account, set it to `FALSE`. The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

XS_AD_RESULT_DATABASE_ENABLED

Category

Analysis & Design

Use to define whether the analysis results database `analysis_results.db5` is created or not. The default setting is `TRUE`. This creates the analysis results database.

If you decide not to create the analysis results database, set this advanced option to `FALSE`. In this way the use of large models with several load combinations is faster and less memory consuming.

This advanced option is model specific and the setting is saved in the options database.

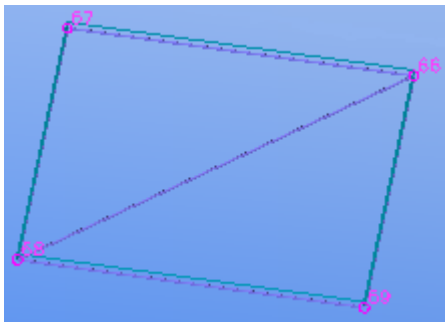
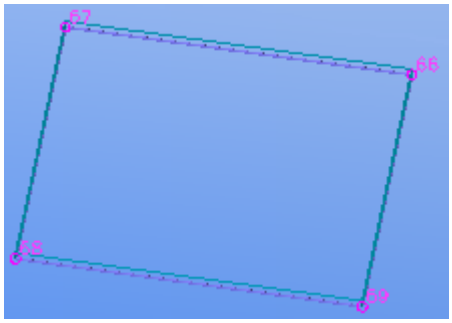
XS_AD_RIGID_DIAPHRAGM_VISUALIZATION

Category

Analysis & Design

Defines whether extra sticks are drawn in visualization for rigid diaphragms.

This advanced option is model specific and the setting is saved in the options database.

Advanced option is set to	Appearance
`TRUE` (default)	
`FALSE`	

XS_AD_SHORT_MEMBER_WARNING_LIMIT

Category

Analysis & Design

Use this advanced option to trigger a warning when a analysis member is shorter that the limit.

Enter the length in millimeters. Default is 50. If the limit is set to 0, no checking is done.

This advanced option is model specific and the setting is saved in the options database.

XS_AD_SHORT_RIGIDLINK_WARNING_LIMIT

Category

Analysis & Design

Use this advanced option to trigger a warning when an analysis rigid link is shorter that the limit. Enter the length in millimeters. Default is `10`. If the limit is set to 0, no checking is done.

This advanced option is model specific and the setting is saved in the options database.

XS_AD_SOLID_AXIAL_EXPAND_MM

Category: Analysis & Design

Used in analysis model creation when checking if parts are clashing. Parts are expanded in axial direction by the given distance to create clash also when there is a small gap between parts. The default value is 25.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_AD_SOLID_SECONDARY_EXPAND_MM (page 70)

XS_AD_SOLID_SECONDARY_EXPAND_MM

Category: Analysis & Design

Used in analysis model creation when checking if parts are clashing and if they should be connected in the analysis model. Secondary parts are expanded in all directions by the given distance to create clash also when there is a gap between parts. The default value is 75.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_AD_SOLID_AXIAL_EXPAND_MM (page 70)

XS_AD_SUPPORT_VISUALIZATION

Category

Analysis & Design

Use to show or hide the support conditions when showing the analysis model in a model view. The default is `TRUE`, which shows the support conditions.

This advanced option is model specific and the setting is saved in the options database.

XS_AD_USE_HIGH_ACCURACY

Category

Analysis & Design

Set this advanced option to `FALSE` to allow for more tolerance in the creation of the analysis model. In certain situations this will result in a more suitable analysis model. If you do not want to allow more tolerance in the analysis mode creation, set this advanced option to `TRUE`. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_ADAPTIVE_OBJECTS

Category

Modeling Properties

Use to define which adaptivity method to use. The options are:

- `ENABLED`: the new adaptivity method is used
- `ENABLED_OLD`: the old adaptivity method is used
- `DISABLED`: adaptivity is disabled

The default value is `ENABLED`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_ADD_SNAPPING_SYMBOL_TO_CIRCLES

Category: Drawing Properties

Set this advanced option to `TRUE` to have the snapping point in the center of circles while you are snapping to the center of circles created in the model by using polygon cuts. You may need to snap to center of circles when you need to create dimensions, for example. The default value is `FALSE`.

If you change the value, you need to reopen the drawing.

This advanced option is model specific and the setting is saved in the options database.

XS_ADJUST_GRID_LABELS

Category

Model View

Set this advanced option to `FALSE` to disable the dynamic moving of grid labels when zooming in, and to keep the labels anchored to the end of the grid line. The default value is `TRUE`, and the grid line labels are kept visible. When working with very large grids having the grid labels always visible might slow down Tekla Structures. After you change the advanced option setting, close and re-open the view to implement the change.

XS_AISC_WELD_MARK

Category: Welds

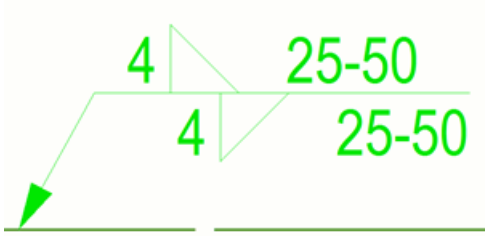
Set this advanced option to `TRUE` to create AISC weld marks, to use leg length as default fillet weld size dimensioning, and to create arrow side weld using below line weld properties.

Set this advanced option to `FALSE` to create ISO weld marks, to use throat thickness as default fillet weld size dimensioning, and to create arrow side weld using above line weld properties. `FALSE` is the default value.

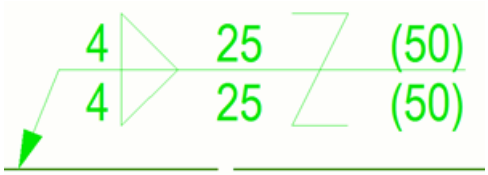
This advanced option also affects the spacing of intermittent welds:

- `TRUE` uses the **Pitch** value entered in the weld properties as the center-to-center spacing of welds.
- `FALSE` uses the **Pitch** value entered in the weld properties as the spacing between the welds.

An AISC weld mark of a staggered intermittent weld with the pitch after a dash:



An ISO weld mark of a staggered intermittent weld with the distance between weld elements in parenthesis



This advanced option is model specific and the setting is saved in the options database.

NOTE The `XS_AISC_WELD_MARK` advanced option is delivered with model folder files. If you need to deliver the model to another party, send the full model with all files (not just the model database file `*.db1`) to ensure that the weld cross section size remains the same. If the value for this advanced option is changed, then the actual weld cross-section size will change accordingly.

XS_ALLOW_DRAWING_TO_MANY_MULTI_DRAWINGS

Category

Numbering

Set this advanced option to `TRUE` to allow the same drawings to be included in more than one multidrawing.

If you want the drawing to be included in only one multidrawing, set this value to `FALSE` (default).

This advanced option is model specific and the setting is saved in the options database.

XS_ALLOW_INCH_MARK_IN_DIMENSIONS

Category

Dimensioning: general

Set to TRUE to have Tekla Structures include the inch symbol in dimensions. The default value is TRUE. If you do not want to allow inch marks, set this advanced option to FALSE.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_ALLOW_INCH_MARK_IN_WELD_SYMBOLS

Category: Imperial Units

Set this advanced option to TRUE to have Tekla Structures include the inch symbol (") in weld symbols. If you do not want to include the inch symbol, set it to FALSE (default).

This advanced option is model specific and the setting is saved in the options database.

XS_ALLOW_REBARS_ON_TOP_OF_EACH_OTHER

Category

Concrete Detailing

If you set this advanced option to TRUE, reinforcing bars can be drawn on top of each other. In this case you have selected **Visibility of reinforcing bars in group** to **bar in the middle of group** or to **customized**, and you have two reinforcing bar groups on top of each other, for example, on the top surface of a slab and on the bottom surface of a slab. The default value is FALSE. This advanced option is model specific and the setting is saved in the options database.

XS_ALLOW_REINFORCING_LOCKED_PARTS

This advanced option must be set in an initialization (.ini) file.

Set to TRUE to create, modify, or delete reinforcement in parts that have been locked. This is useful, for example, when different parties of a project are responsible for modeling concrete parts and for modeling reinforcement, and when modification of parts needs to be prevented.

XS_ALLOW_SHEAR_PLATE_CLASH_FLANGE

Category

Components

Set this advanced option to `TRUE` (default) to omit the default 0.3 mm tolerance between shear tab and secondary part web for shear tab components. To use the tolerance, set it to `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_ALWAYS_CONFIRM_SAVE_WHEN_CLOSING_DRAWING

Category

Drawing Properties

Set this advanced option to `TRUE` to always display the confirmation message "Do you want to save current drawing?".

The default value is `FALSE`. This means that if you close a drawing without making any changes, Tekla Structures does not by default prompt you to save the drawing.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_ALWAYS_CONFIRM_SAVE_WHEN_EXIT

Category

Modeling Properties

Use to define whether or not Tekla Structures prompts you to save the model if you are closing it without making any changes.

If this advanced option is set to `TRUE` (default), Tekla Structures always asks you to save the model when closing it.

If this advanced option is set to `FALSE`, Tekla Structures does not ask you to save the model if there are no changes after the last save.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_ANCHOR_BOLT_PLAN_ADDITIONAL_PARTS_FILTER

Category Drawing properties

Use to add additional parts that are not usually part of a column assembly to an anchor bolt plan, such as anchor rods or leveling plates. First define a general arrangement drawing filter and then give that filter name as the value for this advanced option.

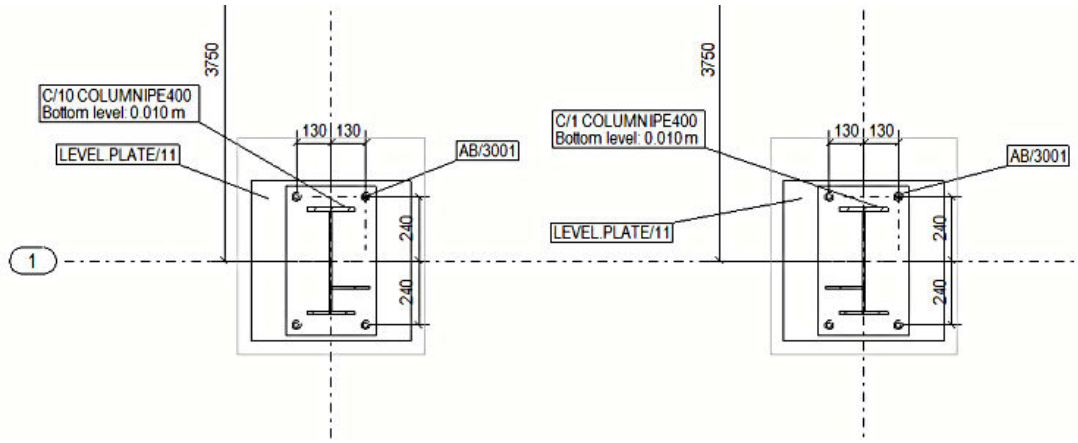
This advanced option is model specific and the setting is saved in the options database.

In the following example we will make anchor rods and leveling plates visible in an anchor bolt plan.

1. Go to the **File** menu and click **Settings** --> **Advanced options** --> **Drawing Properties** .
2. Enter the name of the general arrangement drawing filter for the advanced option `XS_ANCHOR_BOLT_PLAN_ADDITIONAL_PARTS_FILTER`. In this example, we will use the name `SHOW_IN_ABPLAN` (default).
3. On the **Drawings & Reports** tab, click **Drawing properties** > **GA drawing**.
4. In the general arrangement drawing properties dialog box, click **Filter**.
5. Click **Add row**.
6. Select **Part** as the **Category**, **Name** as the **Property**, **Equals** as the **Condition** and enter`*ANCHOR*LEVEL` in the **Value** box.
7. Enter the name of the drawing filter, `SHOW_IN_ABPLAN`, in the **Save as** box and click **Save as**.
8. Click **Cancel** to exit the dialog box.

When you create an anchor bolt plan, both the anchor rods and the leveling plates are shown in the drawing. In the drawing below, the name of the

anchor rod is "ANCHOR ROD" and the name of the leveling plate is "LEVELING PLATE."



XS_ANCHOR_BOLT_PLAN_BASEPLATE_FILTER

Category

Drawing Properties

Use a general arrangement drawing filter to limit the number of visible objects to determine the parts to be included in the anchor bolt plan. First create the base plate filter, then enter its name as the value for the advanced option. Tekla Structures will show in the anchor bolt plan the base plates defined by the drawing filter.

If this advanced option is set, base plate is a part that fulfills the following rules:

- Part belongs to the same assembly as the column, which is the main part of the assembly.
- Part passes the drawing filter specified by `XS_ANCHOR_BOLT_PLAN_BASEPLATE_FILTER`.

If there is more than one part in the column assembly that fulfills the rules, the lowest part is considered to be the base plate.

This advanced option is model specific and the setting is saved in the options database.

Example

```
XS_ANCHOR_BOLT_PLAN_BASEPLATE_FILTER =<the name of the
drawing filter for base plates>
```

XS_ANCHOR_BOLT_PLAN_BOLT_FILTER

Category

Drawing Properties

Use a general arrangement drawing filter to limit the number of visible objects in a drawing and determine the parts to be included in the anchor bolt plan. First create the general arrangement drawing bolt filter, then enter its name as the value for this advanced option. Tekla Structures will show in the anchor bolt plan the bolts included in the drawing filter.

This advanced option is model specific and the setting is saved in the options database.

Example

```
XS_ANCHOR_BOLT_PLAN_BOLT_FILTER =<the name of the drawing
filter for bolts>
```

XS_ANCHOR_BOLT_PLAN_COLUMN_FILTER

Category

Drawing Properties

Use a general arrangement drawing view filter to limit the number of visible objects and determine the parts to be included in the anchor bolt plan. First create a general arrangement drawing column filter, then enter its name as the value for this advanced option. Tekla Structures will show in the anchor bolt plan the columns included in the filter.

This advanced option is model specific and the setting is saved in the options database.

Example

```
XS_ANCHOR_BOLT_PLAN_COLUMN_FILTER =<the name of the drawing
filter for columns>
```

XS_ANCHOR_BOLT_PLAN_DRAWING_TOLERANCE

Category

Drawing Properties

Use to specify the distance of an assembly from the view plane for the assembly to be included in the anchor bolt plan. Enter the distance in millimeters. The default value is `200`.

This advanced option is model specific and the setting is saved in the options database.

XS_ANCHOR_BOLT_PLAN_USE_VIEW_COORDSYS_FOR_BOLT_DIMENSIONS

Category

Drawing Properties

Use to select the direction for the dimensions in the anchor bolt plans. If set to `TRUE`, anchor bolt plan detail view dimensions are created using the coordinate system of the main view. This advanced option is set to `FALSE` by default, which means that the dimensions of anchor bolt plan detail views are created according to the base plate coordinate system.

This advanced option is model specific and the setting is saved in the options database.

XS_ANGLE_DEGREE_SIGN

Category

Drawing Properties

Use to show the degree symbol correctly in Asian languages. By default, the value is `unicode:176`. The value can be changed depending on the desired character. To add the degree symbol in an Asian language, copy the symbol from a text file and paste it directly in the value field. Do not use a unicode value.

This advanced option is model specific and the setting is saved in the options database.

XS_ANGLE_DIMENSION_SYMBOL_SIZE_FACTOR

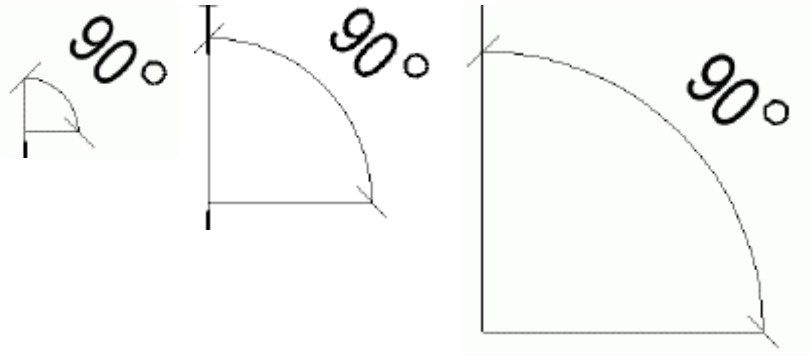
Category

Dimensioning: General

Use to define the size of angle symbols.

This advanced option is model specific and the setting is saved in the options database.

Value 1 Value 3 (default) Value 5



NOTE Setting this advanced option disables the advanced option XS_CHECK_TRIANGLE_TEXT_SIZE.

See also

XS_CHECK_TRIANGLE_TEXT_SIZE (page 118)

XS_ANGLE_TEXT_IN_UNFOLDING_BENDING_LINE_DIMENSIONING

Category: Dimensioning: Unfolding

Use to set the prefix text for unfolded angles. This affects single-part drawings that show folded parts unfolded. Enter any text, for example, A=. By default, there is no prefix text.

This advanced option is model specific and the setting is saved in the options database.

See also

APPL_ERROR_LOG

This advanced option must be set in an initialization (.ini) file.

This advanced option is system specific and is read from teklastructures.ini. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

This advanced option tells the prefix of the application error log files.

Example

If you enter `set APPL_ERROR_LOG=TeklaStructures`, Tekla Structures creates at least the following log files by default in `C:\TeklaStructuresModels\` folder:

- `TeklaStructures_%USERNAME%.log`: log data collected while the program is running
- `TeklaStructures_%USERNAME%.err`: errors collected while the program is running

XS_APPLICATIONS

This advanced option is deprecated, but still supported in Tekla Structures version 2022 and older Tekla Structures versions. Instead of this advanced option, we recommend that you use XS_APPLICATIONS_PATH (page 81), which allows you to define several application file locations.

This advanced option must be set in an initialization (.ini) file.

This advanced option is system specific and is read from `teklastructures.ini`. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

This advanced option defines the folder that contains the connection, detail, and detailing tool applications.

Example

```
set XS_APPLICATIONS=%XS_DIR%\applications\
```

XS_APPLICATIONS_PATH

This advanced option must be set in an initialization (.ini) file.

This advanced option is system specific and is read from `teklastructures.ini`. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

This advanced option defines the folders that contain the connection, detail, and detailing components.

Example

```
set XS_APPLICATIONS_PATH=%XS_APPLICATIONS%;%XSDATADIR
%environments\common\Extensions\applications\
```

The order in which the elements are placed affects how the applications and application settings are used. Tekla Structures reads the first elements first.

XS_ARC_WIDTH_OF_CLOUD

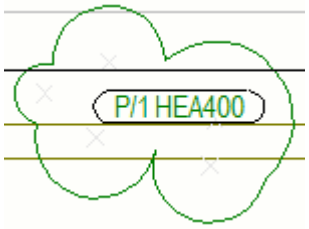
Category

Drawing Properties

Use to control the width of cloud arcs on paper (default 10 mm). You can use the **Cloud** tool to highlight changes in drawings.

This advanced option is model specific and the setting is saved in the options database.

Example

Advanced option is set to	Example of appearance in drawing
1	
10	

XS_ASCII_IMPORT_CREATES_CONSTRUCTION_LINES

Category

Import

Set this advanced option to `TRUE` to have Tekla Structures only draw construction lines between object points in an ASCII import. If you do not want to do this, set it to `FALSE` (default).

This advanced option is model specific and the setting is saved in the options database.

XS_ASSEMBLY_DRAWING_VIEW_TITLE

Category: **Drawing Properties**

Use to define a title for assembly drawing views in a **multidrawing**.

Enter any combination of text and options, and enclose each option in percent symbols (%).

Example: `No. %MODEL_NUMBER%, Marked %ASSEMBLY_POS%`

The options are:

- `PART_NAME`
- `PART_MATERIAL`
- `PART_POS`
- `ASSEMBLY_POS`
- `MODEL_NUMBER`
- `LENGTH`
- `PROFILE`
- `MAIN_PART_LENGTH`
- `DRAWING_BASE_NAME`
- `DRAWING_NAME`

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_ASSEMBLY_FAMILY_POSITION_NUMBER_FORMAT_STRING

Category: Numbering

Use to define the contents of the assembly family position number. For example, instead of DT1-1, DT1-2, you can define the numbering as DT1-A, DT1-B.

To get this result, set the advanced option as follows:

```
XS_ASSEMBLY_FAMILY_POSITION_NUMBER_FORMAT_STRING=
%ASSEMBLY_PREFIX%%ASSEMBLY_FAMILY_NUMBER%-
%ASSEMBLY_FAMILY_QUALIFIER_WITH_LETTERS%
```

This advanced option is model specific and the setting is saved in the options database.

Use the following options to define the contents of family position numbers. Use as many options as you need, and enclose each one in percent symbols (%).

Option	Description
%ASSEMBLY_PREFIX%	Assembly prefix, defined in the part properties in the property pane.
%ASSEMBLY_POS%	The running assembly position number, starting from the start number, defined in the part properties in the property pane.
%ASSEMBLY_FAMILY_NUMBER%	Assembly family number, defined by the start number (in part properties in the property pane) and the final position in that numbering series.
%ASSEMBLY_FAMILY_NUMBER_WITH_LETTERS%	Assembly family number with letters. The letters run automatically from A to Z. If more letters are needed, Tekla Structures takes a second or even a third letter into use, for example AA or AAA. You can also define valid letters with the advanced option XS_VALID_CHARS_FOR_ASSEMBLY_FAMILY_POSITION_NUMBERS.
%ASSEMBLY_FAMILY_QUALIFIER%	Assembly family qualifier, defined by the family numbering criteria given in the **Numbering Setup** dialog box.
%ASSEMBLY_FAMILY_QUALIFIER_WITH_LETTERS%	Assembly family qualifier with letters. The letters run automatically from A to Z. If more letters are needed, Tekla Structures takes a second or even a third letter into use, for example AA or AAA. You can also define valid letters with the advanced option XS_VALID_CHARS_FOR_ASSEMBLY_FAMILY_QUALIFIER.
%UDA:[uda_name]%	User-defined attribute of the assembly or assembly main part. If the attribute is defined for the assembly, it will be used. If it is not defined, the main part attribute will be used. You can also define that only the user-defined attribute of the main part will be used, for example %UDA:MAINPART.USER_FIELD_1%.

Option	Description
%TPL:[tpl_name]%	Template attribute of the assembly or assembly main part, for example %TPL:PROJECT.NUMBER% .

Number of characters

You can define the number of characters by adding a period and the required number to the end of each option. For example %ASSEMBLY_FAMILY_QUALIFIER_WITH_LETTERS.3% will result in three letters for every assembly, starting from AAA.

Example

If you set the advanced option to

```
XS_ASSEMBLY_FAMILY_POSITION_NUMBER_FORMAT_STRING=
%ASSEMBLY_PREFIX%/%ASSEMBLY_FAMILY_NUMBER.3%-
%ASSEMBLY_FAMILY_QUALIFIER.3%
```

The result will be

```
A/001-001.
```

See also

ASSEMBLY_POS (page 515)

XS_VALID_CHARS_FOR_ASSEMBLY_FAMILY_POSITION_NUMBERS (page 491)

XS_VALID_CHARS_FOR_ASSEMBLY_FAMILY_QUALIFIER (page 491)

Template attributes in drawing and report templates (page 505)

XS_ASSEMBLY_MULTI_NUMBER_FORMAT_STRING

Category: Numbering

Use to base assembly numbers on drawing numbers. Use the following switches to define the contents of assembly marks. Use as many switches as you need, and enclose each one in percent symbols (%).

The available switches are:

Switch	Description
%ASSEMBLY_MULTI_DRAWING_NUMBER%	Multidrawing name.
%ASSEMBLY_MULTI_DRAWING_POS%	Position of the assembly drawing inside the multidrawing.
%ASSEMBLY_PREFIX%	Assembly prefix in the model.

Switch	Description
%ASSEMBLY_POS%	Assembly position number in the model.
Template fields	Enter TPL: followed by the name of any relevant template field. Enclose each name in percent symbols (%). For example, %TPL:PROJECT.NUMBER% **NOTE:** No template fields starting with DRAWING work for this advanced option, for example, DRAWING.TITLE1. If you use them in the value, Tekla Structures will replace the whole value string with the default value.
User-defined attributes that are defined in the objects.inp file	Enter UDA: followed by the name of any relevant user-defined attribute, exactly as it appears in the objects.inp file. For example, %UDA:MY_INFO_1%

This advanced option is model specific and the setting is saved in the options database.

Example

To put the assembly multinumbers in the format multidrawing name + part prefix + position on multidrawing, set the advanced option as follows:

```
%ASSEMBLY_MULTI_DRAWING_NUMBER%%ASSEMBLY_PREFIX%
%ASSEMBLY_MULTI_DRAWING_POS%
```

This will create the assembly mark 10B1, where:

- 10 is the drawing number
- B is the assembly prefix
- 1 designates that it is the first assembly on the sheet.

See also

XS_USE_MULTI_NUMBERING_FOR (page 472)

XS_PART_MULTI_NUMBER_FORMAT_STRING (page 340)

XS_CAST_UNIT_MULTI_NUMBER_FORMAT_STRING (page 109)

XS_ASSEMBLY_POSITION_CODE_3D

Category

Templates and symbols

Set to `TRUE` to include Z orientation in assembly position codes.

The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_ASSEMBLY_POSITION_CODE_TOLERANCE

Category

Templates and symbols

Tolerance distance to use when identifying grid position code for assembly. The default value is `500.0`.

This advanced option is model specific and the setting is saved in the options database.

XS_ASSEMBLY_POSITION_NEW_FORMAT

Category

Templates and Symbols

Use to define how the assembly position code is displayed in reports. Set to `TRUE` to use a format such as `B1/C2` instead of `B-C/1-2`. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_ASSEMBLY_POSITION_NUMBER_FORMAT_STRING

Category: Numbering

Use to define the contents of the assembly position number.

NOTE Do not change this advanced option after you have created single-part, assembly, or cast unit drawings, if you are using cloning. It may partially separate the drawings from the part they represent, causing

the drawings to be flagged as deleted and other drawings to be cloned after the next numbering.

NOTE The advanced options XS_ASSEMBLY_POSITION_NUMBER_FORMAT_STRING and XS_PART_POSITION_NUMBER_FORMAT_STRING (page 340) override the advanced option XS_SWITCH_POS_NUMBERS_FOR (page 445). XS_SWITCH_POS_NUMBERS_FOR has no impact on assembly and/or part position numbers if you use XS_ASSEMBLY_POSITION_NUMBER_FORMAT_STRING and/or XS_PART_POSITION_NUMBER_FORMAT_STRING.

The options are:

Option	Description
%ASSEMBLY_PREFIX%	Assembly prefix, defined in the part properties dialog box.
%ASSEMBLY_POS%	The running assembly position number, starting from the start number defined in the part properties dialog box.
%ASSEMBLY_POS_WITH_LETTERS%	Same as above, but with letters. Uses letters A – Z by default, but you can also define valid letters with the advanced option XS_VALID_CHARS_FOR_ASSEMBLY_POSITION_NUMBERS.
%UDA:[uda_name]%	User-defined attribute of the assembly or assembly main part. If the attribute is defined for the assembly, it will be used. If it is not defined, the main part attribute will be used. You can also define that only the user-defined attribute of the main part will be used, for example %UDA:MAINPART.USER_FIELD_1%.
%TPL:[tpl_name]%	Template attribute of the assembly or assembly main part, for example %TPL:PROJECT.NUMBER% .

This advanced option is model specific and the setting is saved in the options database.

Number of characters in the assembly position number

You can define the number of characters by adding a period and the required number to the end of the options `%ASSEMBLY_POS_WITH_LETTERS%` and `%ASSEMBLY_POS%`. For example,`%ASSEMBLY_POS_WITH_LETTERS.3%` will result in three letters for every assembly, starting from `AAA`.

Example

If you set the advanced option to `%ASSEMBLY_PREFIX%/%ASSEMBLY_POS.3%`, the result will be `A/001`.

See also

ASSEMBLY_POS (page 515)

XS_VALID_CHARS_FOR_ASSEMBLY_POSITION_NUMBERS (page 492)

XS_ASSOCIATIVE_CHANGE_HIGHLIGHT_SIZE

Category

Dimensioning: General

Use to define the height of the change symbol around the points. The default value is `7`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_HIGHLIGHT_ASSOCIATIVE_DIMENSION_CHANGES (page 277)

XS_ASSOCIATIVE_CHANGE_HIGHLIGHT_SYMBOL

Category

Dimensioning: General

Use to define the used associativity change symbol. The default value is `AssociativityAnchor@1` (a cloud).

This advanced option is model specific and the setting is saved in the options database.

See also

XS_HIGHLIGHT_ASSOCIATIVE_DIMENSION_CHANGES (page 277)

XS_ATTRIBUTE_FILE_EXCLUDE_LIST

Category

File Locations

Use this advanced option to exclude attribute files or reports from the user interface. For this advanced option control strings are given for controlling the access to attribute files or reports. If a control string is present anywhere in a filename of an attribute file or a report, the file or report becomes unavailable. Several control strings can be entered, and the values are case sensitive. Use semicolon (;) as a separator.

The default value is
`TS_Report_Inquire;dgn_attribute_info;import_revision`

- Enter the file name or a part of the file name that you want to exclude from the user interface as the value for the advanced option. For example, setting the advanced option to `_eng;_det` removes all the attribute files that include the string `_eng` or `_det` in their name from the user interface.
- Enter the name or a part of the name of the report you want to exclude from the **Create report** dialog box. For example, setting the advanced option to `_Part` hides the report `ts_Report_Inquire_Part.rpt` in the dialog box.

Example:

```
PROJ1_;PROJ2_;_TeamA;Team_Detailing
```

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_AUTOCONNECTION_TOLERANCE

Category: Components

Use to set the size of the area that Tekla Structures searches for parts to connect with **AutoConnection**. Enter a value in millimeters, for example, `500`. The default value is `0`, which means that the connection is created only between colliding or adjacent parts.

This advanced option is model specific and the setting is saved in the options database.

XS_AUTOCONNECTION_USE_UDL

Category

Components

Set this advanced option to `TRUE` to switch on UDL calculation in **AutoConnection**. This calculates the maximum shear force allowed. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_AUTODEFAULT_UDL_PERCENT

Category

Components

Use to set a default percentage for UDL calculation. The default value is `50`.

In **AutoDefaults**Tekla Structures uses the percentage in the connection properties. If no percentage is given, Tekla Structures uses this value.

In **AutoConnection** only the default percentage is used. You switch UDL calculation on with the advanced option `XS_AUTOCONNECTION_USE_UDL`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_AUTOCONNECTION_USE_UDL (page 90)

XS_AUTOMATIC_NEW_MODEL_NAME

Category

Modeling Properties

Set this advanced option to `TRUE` (default) to have Tekla Structures suggest a name for a new model automatically in the **New** dialog box. The suggested name is `New model XX`, where `XX` is a running number.

If you set this advanced option to `FALSE`, the name is not suggested.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_AUTOMATIC_USER_FEEDBACK_SAVING_INTERVAL

Category

Modeling Properties

Enter an integer to define the saving interval of the usage statistics information. The saving interval means the number of commands or interactions that the data collection observes before saving data to file. The value of the advanced option corresponds to the number of rows in the log file. The default value is `100`.

The log file is called `UserFeedbackLog.txt`, and it is located in the `Logs` folder under the path defined with **XSUSERDATADIR** in `teklastructures.ini` file.

See also

XS_AUTOMATIC_USER_FEEDBACK_SENDING_INTERVAL (page 92)

XSUSERDATADIR (page 489)

XS_AUTOMATIC_USER_FEEDBACK_SENDING_INTERVAL

Category

Modeling Properties

Enter an integer to define the sending interval of usage statistics information. The value of the advanced option corresponds to the number of rows in the log file. The default value is `10000`.

The log file is called `UserFeedbackLog.txt`, and it is located in the `Logs` folder under the path defined with **XSUSERDATADIR** in `teklastructures.ini` file.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_AUTOMATIC_USER_FEEDBACK_SAVING_INTERVAL (page 92)

XSUSERDATADIR (page 489)

XS_AUTOSAVE_DIRECTORY

Category

File Locations

Enter the path to the folder that contains the files that Tekla Structures saves automatically. The default value is `%XS_RUNPATH%\autosave\`.

If you do not define a path, Tekla Structures places autosave files in the current model folder.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

1.2 Advanced options - B

XS_BACKGROUND_COLOR1

Category: Model View

To have a gradient background color in model views, use the advanced options XS_BACKGROUND_COLOR1 (page 93), XS_BACKGROUND_COLOR2 (page 94), XS_BACKGROUND_COLOR3 (page 94), and XS_BACKGROUND_COLOR4 (page 94). With these advanced options you can control the color of each corner separately in a model view.

This advanced option controls the upper-left corner of the model view.

Define the color using RGB (Red Green Blue) values. The scale is from 0 to 1, and separate the numbers with spaces. If you want to use the traditional single-colored background, set the same value for all four background color advanced options. Reopen the view for the change to take effect.

The default value is `1.0 1.0 1.0`. To use the default background color, leave the box empty.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`.

Example

- Use `0.0 0.0 0.0` for black background.
- Use `1.0 1.0 1.0` for white background.

XS_BACKGROUND_COLOR2

Category: Model View

See XS_BACKGROUND_COLOR1 (page 93).

This advanced option controls the upper-right corner of the model view.

The default value is `1.0 1.0 1.0`. To use the default background color, leave the box empty.

XS_BACKGROUND_COLOR3

Category: Model View

See XS_BACKGROUND_COLOR1 (page 93).

This advanced option controls the bottom-left corner of the model view.

The default value is `0.80 0.79 0.85`. To use the default background color, leave the box empty.

XS_BACKGROUND_COLOR4

Category: Model View

See XS_BACKGROUND_COLOR1 (page 93).

This advanced option controls the bottom-right corner of the model view.

The default value is `0.80 0.79 0.85`. To use the default background color, leave the box empty.

XS_BASE_LINE_WIDTH

Category

Plotting

Use this advanced option to specify the base line width for printed drawings. Enter the value in millimeters as decimal value. The default value is `0.01`. The final line thicknesses in a printed drawing is the base line width multiplied by the pen thickness from the **Print Drawings** dialog box (or in the old printing, from the **Color** dialog box). For example, `25` will give a thickness of `0.25` mm.

TIP To control the accuracy of line thickness, use a small value for `XS_BASE_LINE_WIDTH` and a large number for the pen.

This advanced option also affects lines on the screen, if you set `XS_BASE_LINE_WIDTH_AFFECTS_SCREEN` to `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_BASE_LINE_WIDTH_AFFECTS_SCREEN (page 95)

XS_BASE_LINE_WIDTH_AFFECTS_SCREEN

Category

Plotting

Set this advanced option to `TRUE`, if you want `XS_BASE_LINE_WIDTH`to affect line widths drawn on the screen.

Enter `FALSE` if you do not want `XS_BASE_LINE_WIDTH` to affect line widths on the screen.

The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_BASE_LINE_WIDTH (page 94)

XS_BASICVIEW_HEIGHT

Category

Model View

Use to define the height of basic view windows. Enter the height in pixels. The default value is `375`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

Example

```
XS_BASICVIEW_HEIGHT=570
```

See also

XS_BASICVIEW_POSITION_X (page 96)

XS_BASICVIEW_POSITION_X

Category

Model View

Use to control the horizontal position of basic view windows on the screen. Enter the value in pixels. The default value is `100`.

If the advanced option `XS_MDIBASICVIEWPARENT` is set, the origin for the position is the top left corner of the client area. Otherwise, the origin is the top left corner of the whole Tekla Structures window. Moving a toolbar also affects the size of the client area, because menus and toolbars are not part of it.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_MDIBASICVIEWPARENT (page 315)

XS_BASICVIEW_POSITION_Y

Category

Model View

Use to control the vertical position of basic view windows on the screen. Enter the value in pixels. The default value is `20`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_BASICVIEW_POSITION_X (page 96)

XS_BASICVIEW_WIDTH

Category

Model view

Use to define the width of basic view windows. Enter the width in pixels, for example, 570. The default value is `375`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_BASICVIEW_POSITION_X (page 96)

XS_BEVEL_DIMENSIONS_FOR_PROFILES_ONLY

Category

Dimensioning: Parts

Use to define whether to show bevel dimensions only for profiles.

If you set the advanced option to `TRUE`, bevel dimensions are shown only for profiles

If you set the advanced option to `FALSE` (default), bevel dimensions are shown always.

This advanced option is model specific and the setting is saved in the options database.

XSBIN

This advanced option must be set in an initialization (.ini) file.

This advanced option is system specific and is read from `teklastructures.ini`. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

This advanced option defines the location of the Tekla Structures `bin` folder.

XS_BLACK_DRAWING_BACKGROUND

Category: Drawing View

Set to `TRUE` to have a black background and colored lines in drawings. If you want to have white background and colored lines, enter `FALSE` (default).

This advanced option only affects the **Color** mode, not the **Black and White** or **Gray Scale** mode.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`.

Note that you do not need to restart to activate the changed value, just reopen the drawing.

See also

XS_USE_COLOR_DRAWINGS (page 463)

XS_BOLT_DUPLICATE_IGNORE

Category: Numbering

NOTE Do not use this advanced option in new models. This advanced option was a temporary solution that has been replaced with a more comprehensive option.

Instead of the `XS_BOLT_DUPLICATE_...` advanced options, use the **No hole** option for **Special hole type** in the **Bolt** properties.

If necessary in existing models, use this advanced option to define whether duplicate bolt holes in the same location, or within a tolerance (page 99), in a part are ignored in part numbering, and in drawings and NC files.

The default value is `FALSE`.

If this advanced option is set to `TRUE`, the duplicate bolt holes are ignored in part numbering, and in drawings and NC files.

You can use this advanced option when only some and possibly a different set of the holes in otherwise identical parts will be used for bolting on site. In this way identical parts can be used in different conditions in the structure.

The size of the duplicate holes is not taken into consideration, only the location, so holes of different size are considered duplicates if they are in the same location.

If there are overlapping holes within the tolerance, the hole without a bolt is taken into account, and the hole created by an actual bolt is ignored.

NOTE It is not recommended to change this advanced option during a project, but if you do so, you must carry out full numbering.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_BOLT_DUPLICATE_TOLERANCE (page 99)

XS_BOLT_DUPLICATE_TOLERANCE

Category: Numbering

NOTE Do not use this advanced option in new models. This advanced option was part of a temporary solution that has been replaced with a more comprehensive option.

Instead of the `XS_BOLT_DUPLICATE_...` advanced options, use the **No hole** option for **Special hole type** in the **Bolt** properties.

If necessary in existing models, when XS_BOLT_DUPLICATE_IGNORE (page 98) is set to `TRUE`, use `XS_BOLT_DUPLICATE_TOLERANCE` to set the location tolerance that defines which bolt holes are considered duplicates and will therefore be ignored in part numbering.

The default value for the tolerance is 3 mm.

NOTE It is not recommended to change this advanced option during a project, but if you do so, you must carry out full numbering.

This advanced option is model specific and the setting is saved in the options database.

XS_BOLT_LENGTH_EPSILON

Category

Modeling Properties

Use to round bolt length.

Tekla Structures calculates bolt length using material thickness. To avoid having several bolt lengths where material thickness only differs slightly, set this advanced option to a positive or negative value. This value is multiplied by two and the result is subtracted from the material thickness in bolt length

calculation. Enter the value in millimeters (all environments). Typical values are 0.001–0.5. The default value is 0.001.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_BOLT_MARK_DIAMETER_PREFIX

Category

Marking: Bolts

Use this advanced option to specify a prefix for bolt mark diameter. The possible values are `NONE` or any string.

This advanced option is model specific and the setting is saved in the options database.

Example

For a bolt group consisting of three 20 mm diameter bolts:

- If you do not give any value, result is

  3*M20
- If you set `XS_BOLT_MARK_DIAMETER_PREFIX` to `NONE`, the result is

  3*20
- If you set `XS_BOLT_MARK_DIAMETER_PREFIX` to `D`, the result is

  3*D20

XS_BOLT_MARK_IS_ALWAYS_VISIBLE

Category

Marking: Bolts

Set this advanced option to `TRUE` to show the bolt marks of bolts hidden from views by other objects. The bolt marks are displayed with solid leader lines and frames. If you set it to `FALSE` (default), the bolt marks of hidden bolts are displayed with dashed leader lines and frames. This advanced option affects all drawing types.

This advanced option is model specific and the setting is saved in the options database.

XS_BOLT_MARK_IS_ALWAYS_VISIBLE_IN_GA

Category

Marking: Bolts

Set this advanced option to `TRUE` to show the bolt marks of bolts hidden from views by other objects. The bolt marks are displayed with solid leader lines and frames. If you set it to `FALSE` (default), the bolt marks of hidden bolts are displayed with dashed leader lines and frames. This advanced only affects general arrangement drawings.

NOTE If you have set the advanced option `XS_BOLT_MARK_IS_ALWAYS_VISIBLE` to `TRUE`, the bolt marks are always displayed with solid lines even if you set `XS_BOLT_MARK_IS_ALWAYS_VISIBLE_IN_GA` to `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_BOLT_MARK_STRING_FOR_SIZE

Category

Marking: bolts

Use to define the contents of the **Size** element in bolt marks. For example, to have the bolt number and the hole diameter in the mark, enter `%BOLT_NUMBER%*D%HOLE.DIAMETER%`.

This advanced option is only used when there is a bolt and the hole is a normal one.

The advanced options `XS_SHOP_BOLT_MARK_STRING_FOR_SIZE` and `XS_SITE_BOLT_MARK_STRING_FOR_SIZE` override this setting.

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER

- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)
- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD
- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK
- BOLT_SHORT_NAME
- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SITE_BOLT_MARK_STRING_FOR_SIZE (page 431)

XS_SHOP_BOLT_MARK_STRING_FOR_SIZE (page 409)

XS_BOLT_MARK_STRING_FOR_SIZE_IN_GA

Category

Marking: Bolts

Use to define the contents of the size element in bolt marks in general arrangement drawings. If you have not set the advanced options `XS_SHOP_BOLT_MARK_STRING_FOR_SIZE_IN_GA` or `XS_SITE_BOLT_MARK_STRING_FOR_SIZE_IN_GA`, then this advanced option is used.

This advanced option is only used when there is a bolt and the hole is a normal one.

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER

- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)
- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD
- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK
- BOLT_SHORT_NAME
- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SHOP_BOLT_MARK_STRING_FOR_SIZE_IN_GA (page 410)

XS_SITE_BOLT_MARK_STRING_FOR_SIZE_IN_GA (page 432)

XS_BOLT_POSITION_TO_MIN_AND_MAX_POINT

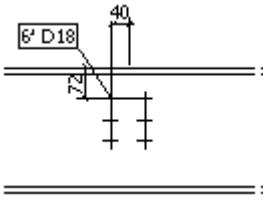
Category

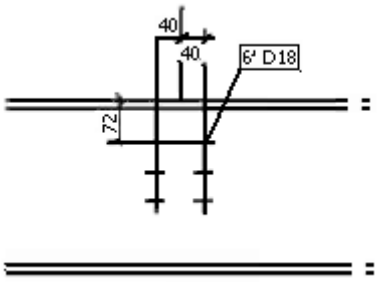
Dimensioning: Bolts

Set this advanced option to `TRUE` to create minimum and maximum position dimensions for bolts. If you do not want to create minimum and maximum position dimensions for bolts, set it to `FALSE` (default).

This advanced option is model specific and the setting is saved in the options database.

The examples below show a maximum bolt dimension of 40:

Setting	Example
Before setting the advanced option.	

Setting	Example
After setting the advanced option to TRUE.	

See also

XS_BOLT_REPRESENTATION_SYMBOL_AXIS_POSITION_AS_EXACT_SOLID

Category

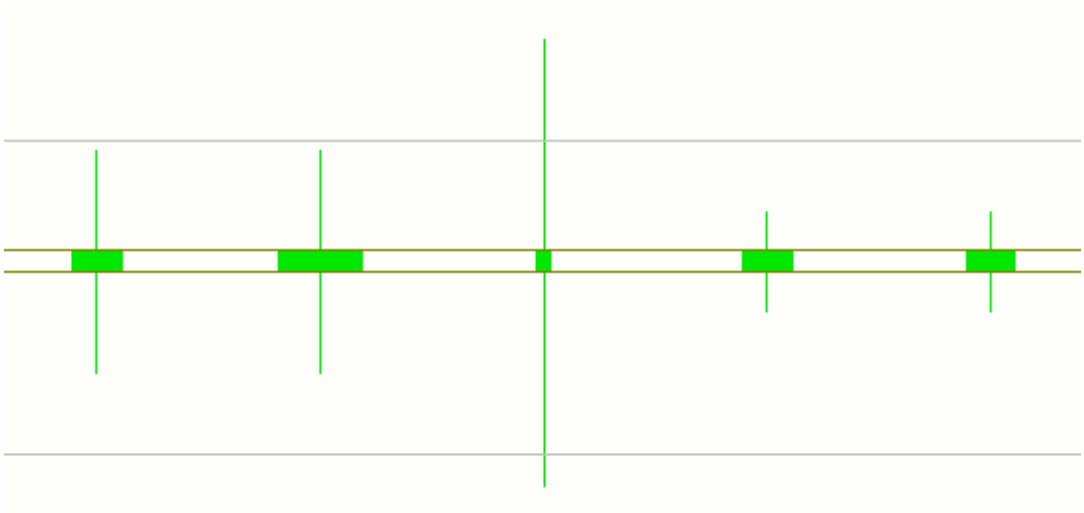
Drawing Properties

If you set the advanced option XS_BOLT_REPRESENTATION_SYMBOL_AXIS_POSITION_AS_EXACT_SOLID to TRUE, all bolt axis representations set to **Symbol** are presented in the same position as with **Exact solid**.

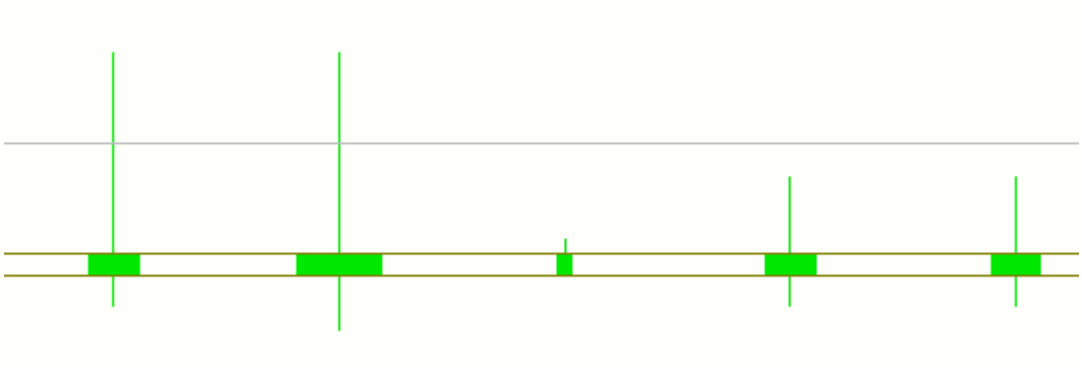
The default value is TRUE.

This advanced option is model specific and the setting is saved in the options database.

Symbol representation when the advanced option is set to FALSE:



Symbol representation when the advanced option is set to TRUE:



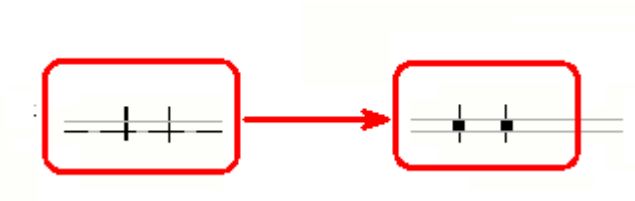
See also

XS_BOLT_REPRESENTATION_USE_POSITIVE_CUT_LENGTH

Category

Drawing Properties

Set this advanced option to `TRUE` to create bolt holes in the drawing instead of sticks when the cut length is negative. `FALSE` is the default value.



This advanced option is model specific and the setting is saved in the options database.

XS_BOLTS_PERPENDICULAR_TO_PART_PLANE_IN_NC

Category

CNC

Set this advanced option to `TRUE` (default) to dimension bolts perpendicular to part plane in NC files. If you do not want to dimension the bolts, set it to `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

1.3 Advanced options - C

XS_CALCULATE_POLYBEAM_LENGTH_ALONG_REFERENCE_LINE

Category

Dimensioning: Unfolding

Use this advanced option to calculate polybeam length along the reference line instead of the centerline. By default, Tekla Structures measures polybeam length along the centerline of the beam, regardless of the **Position in plane** values of the beam.

Set this advanced option to `TRUE` to calculate polybeam length along the reference line. Set it to `FALSE` (default) to calculate along the centerline.

When this advanced option is set, Tekla Structures ignores all unfolding parameters defined in the `unfold_corner_ratios.inp` file. However, this only affects polybeams with straight sections. The length of polybeams with curved chamfers is always controlled using `unfold_corner_ratios.inp`.

This advanced option is model specific and the setting is saved in the options database.

NOTE The centerline option is always a safe alternative, because other modifications do not impact on it. For example, if you calculate length using the reference line and you have used offsets to move the beam, the result may not be the true length of the beam.

NOTE When XS_USE_OLD_POLYBEAM_LENGTH_CALCULATION (page 477) is set to `TRUE`, this advanced option is not used, even though results might be the same in most cases. To obtain reliable results, especially for Net Length, we recommend that you do not set `XS_USE_OLD_POLYBEAM_LENGTH_CALCULATION` to `TRUE`.

See also

XS_CALCULATE_POUR_UNITS_ON_SHARING

Category: Multi-user

When this advanced option is set to `FALSE` (which is the default value), each user has to run the **Calculate pour units** command in their local version of the shared model to update the pour units.

If this advanced option is set to TRUE, pour units are automatically calculated and up to date in the shared model after read in and write out.

This advanced option is model specific and the setting is saved in the options database.

XS_CAST_UNIT_FAMILY_POSITION_NUMBER_FORMAT_STRING

Category

Numbering

Use to define the contents of the cast unit family position number. For example, instead of DT1-1, DT1-2, you can define the numbering as DT1-A, DT1-B.

To get this result, set the advanced option as follows:

```
%CAST_UNIT_PREFIX%%CAST_UNIT_FAMILY_NUMBER%-
%CAST_UNIT_FAMILY_QUALIFIER_WITH_LETTERS%
```

Use the following options to define the contents of family position numbers. Use as many options as you need, and enclose each one in percent symbols (%).

Option	Description
%CAST_UNIT_PREFIX%	Cast unit prefix, defined in the part properties in the property pane.
%CAST_UNIT_POS%	The running cast unit position number, starting from the start number defined in the part properties in the property pane.
%CAST_UNIT_FAMILY_NUMBER%	Cast unit family number, defined by the start number (in part properties in the property pane) and the final position in that numbering series.
%CAST_UNIT_FAMILY_NUMBER_WITH_LETTERS%	Cast unit family number with letters. The letters run automatically from A to Z. If more letters are needed, Tekla Structures takes a second or even a third letter into use, for example AA or AAA. You can also define valid letters with the advanced option XS_VALID_CHARS_FOR_ASSEMBLY_FAMILY_POSITION_NUMBERS.

Option	Description
%CAST_UNIT_FAMILY_QUALIFIER%	Cast unit family qualifier, defined by the family numbering criteria given in the **Numbering Setup** dialog box.
%CAST_UNIT_FAMILY_QUALIFIER_WITH_LETTERS%	Cast unit family qualifier with letters. The letters run automatically from A to Z. If more letters are needed, Tekla Structures takes a second or even a third letter into use, for example AA or AAA. You can also define valid letters with the advanced option XS_VALID_CHARS_FOR_ASSEMBLY_FAMILY_QUALIFIER.
%UDA:[uda_name]%	User-defined attribute of the cast unit or cast unit main part. If the attribute is defined for the cast unit, it will be used. If it is not defined, the main part attribute will be used. You can also define that only the user-defined attribute of the main part will be used, for example %UDA:MAINPART.USER_FIELD_1%.
%TPL:[tpl_name]%	Template attribute of the cast unit or cast unit main part, for example %TPL:PROJECT.NUMBER%.

This advanced option is model specific and the setting is saved in the options database.

Number of characters

You can define the number of characters by adding a period and the required number to the end of each option. For example %CAST_UNIT_FAMILY_QUALIFIER_WITH_LETTERS.3% will result in three letters for every cast unit, starting from AAA.

Example

If you set the advanced option to

```
%CAST_UNIT_PREFIX%/%CAST_UNIT_FAMILY_NUMBER.3%-
%CAST_UNIT_FAMILY_QUALIFIER.3%
```

The result will be

```
A/001-001.
```

See also

CAST_UNIT_POS (page 524)

XS_VALID_CHARS_FOR_ASSEMBLY_FAMILY_QUALIFIER (page 491)

XS_VALID_CHARS_FOR_ASSEMBLY_FAMILY_POSITION_NUMBERS (page 491)

XS_CAST_UNIT_MULTI_NUMBER_FORMAT_STRING

Category: Numbering

Use to base cast unit numbers on drawing numbers. Use the following switches to define the contents of cast unit marks. Use as many switches as you need, and enclose each one in percent symbols (%).

The available switches are:

Switch	Description
`%CAST_UNIT_MULTI_DRAWING_NUMBER%`	Multidrawing name.
`%CAST_UNIT_MULTI_DRAWING_POS%`	Position of the cast unit drawing inside the multidrawing.
`%CAST_UNIT_PREFIX%`	Cast unit prefix in the model.
`%CAST_UNIT_POS%`	Cast unit position number in the model.
Template fields	Enter `TPL:` followed by the name of any relevant template field. Enclose each name in percent symbols (%). For example, `%TPL:PROJECT.NUMBER%` **NOTE:** No template fields starting with `DRAWING` work for this advanced option, for example, `DRAWING.TITLE1`. If you use them in the value, Tekla Structures will replace the whole value string with the default value.
User-defined attributes that are defined in the `objects.inp` file	Enter `UDA:` followed by the name of any relevant user-defined attribute, exactly as it appears in the `objects.inp` file. For example, `%UDA:MY_INFO_1%`

This advanced option is model specific and the setting is saved in the options database.

Example

To put the cast unit multinumbers in the format multidrawing name + part prefix + position on multidrawing, set the advanced option as follows:

```
%CAST_UNIT_MULTI_DRAWING_NUMBER%%CAST_UNIT_PREFIX%
%CAST_UNIT_MULTI_DRAWING_POS%
```

This will create the cast unit mark 10B1, where:

- 10 is the drawing number
- B is the cast unit prefix
- 1 designates that it is the first cast unit on the sheet.

See also

XS_USE_MULTI_NUMBERING_FOR (page 472)

XS_PART_MULTI_NUMBER_FORMAT_STRING (page 340)

XS_ASSEMBLY_MULTI_NUMBER_FORMAT_STRING (page 85)

XS_CAST_UNIT_POSITION_NUMBER_FORMAT_STRING

Category: Numbering

Use to define the contents of the cast unit position number.

NOTE Do not change this advanced option after you have created single-part, assembly, or cast unit drawings, if you are using cloning. It may partially separate the drawings from the part they represent, causing the drawings to be flagged as deleted and other drawings to be cloned after the next numbering.

The options are:

Option	Description
%CAST_UNIT_PREFIX%	Cast unit prefix, defined in the part properties dialog box.
%CAST_UNIT_POS%	The running cast unit position number, starting from the start number defined in the part properties dialog box.
%CAST_UNIT_POS_WITH_LETTERS%	Same as above, but with letters. Uses letters A – Z by default, but you can also define valid letters with the advanced option XS_VALID_CHARS_FOR_ASSEMBLY_POSITION_NUMBERS

Option	Description
%UDA:[uda_name]%	User-defined attribute of the cast unit or cast unit main part. If the attribute is defined for the cast unit, it will be used. If it is not defined, the main part attribute will be used. You can also define that only the user-defined attribute of the main part will be used, for example %UDA:MAINPART.USER_FIELD_1%.
%TPL:[tpl_name]%	Template attribute of the cast unit or cast unit main part, for example %TPL:PROJECT.NUMBER%.

This advanced option is model specific and the setting is saved in the options database.

Number of characters

You can define the number of characters by adding a period and the required number to the end of the options %CAST_UNIT_POS% and %CAST_UNIT_POS_WITH_LETTERS%. For example, %CAST_UNIT_POS_WITH_LETTERS.3% will result in three letters for every cast unit, starting from AAA.

Example

If you set the advanced option to %CAST_UNIT_PREFIX%/%CAST_UNIT_POS.3%, the result will be A/001.

See also

CAST_UNIT_POS (page 524)

XS_CENTER_LINE_TYPE

Category: Drawing Properties

Drawing Properties

Use this advanced option to change the line type of the part center lines in drawings. Enter an integer value from 1 to 7. To see the center line type change, close the drawing and open it again.

You can select whether to show the center line in Part properties (page 759).

Value	Description
1	Solid line
2 – 7	Dotted lines. The default value is 4 (dash-and-dot line).

If you enter a value that is lower than 1 or higher than 7, Tekla Structures uses the default value 4.

TIP To see what the line types look like, see the **Hidden lines** > **Type** options on the **Appearance** tab in **Part Properties**.

For a list of default line types, see .

This advanced option is model specific and the setting is saved in the options database.

XS_CENTER_TO_CENTER_DISTANCE_IN_ONE_PART_STRING

Category

Marking: Bolts

Sets the format of the element **Center-to-center distance** in bolt marks, when the bolts are in the same part, for example, in a column or beam flange. Use to define gage information in the bolt mark in the following way:

- Use the switch %VALUE% for the numeric value of a gage.
- You can add text and the option in any order, for example, %VALUE% GAGE or GAGE %VALUE%.
- If %VALUE% is missing, Tekla Structures adds the center-to-center distance to the end of the string.
- If this advanced option is not set, only %VALUE% is used.
- If Tekla Structures cannot calculate the gage, nothing is added in the mark.
- The default value value is GAGE = %VALUE% .

This advanced option is model specific and the setting is saved in the options database.

Example

There is a symmetrical bolt group in a beam flange. The value for gage is 10.

In the Advanced Options dialog box	In bolt mark
`GAGE = %VALUE%`	GAGE = 10

XS_CENTER_TO_CENTER_DISTANCE_IN_TWO_PARTS_STRING

Category

Marking: Bolts

Sets the format of the element **Center-to-center distance** in bolt marks where there are two parts.

The default value is `C/C = %VALUE%`.

This advanced option is model specific and the setting is saved in the options database.

Example

There is a symmetrical bolt group in two clip angles. The center-to-center distance is 10.

In the Advanced Options dialog box	In bolt mark
`c/c = %VALUE%`	c/c = 10

See also

XS_CENTER_TO_CENTER_DISTANCE_IN_ONE_PART_STRING (page 112)

XS_CHAMFER_ACCURACY_FACTOR

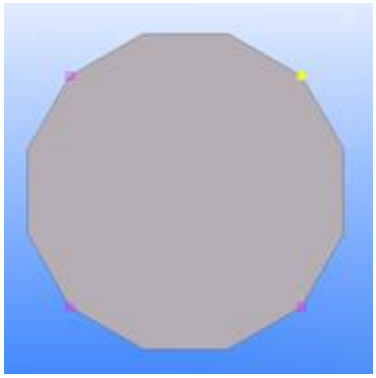
Category: Speed and accuracy

Use this advanced option to define the number of points used for chamfers. Use lower values to increase the number of points used to construct the chamfer, thus making the curved chamfer smoother. Use larger values to decrease the number of points used to construct the chamfer, thus making the curved chamfer less smoother.

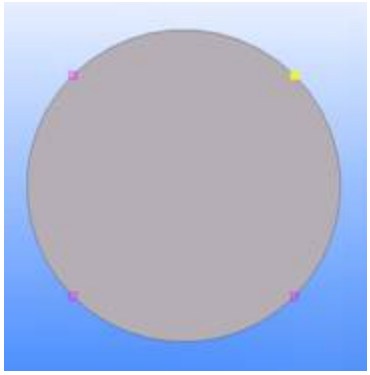
The default value is `4.0`.

Restart Tekla Structures after changing the value to activate the new setting.

In the example below, the value is 16.



In the example below, the value is 1.



The minimum value is 0.1.

Click the links below to find out more:

Round chamfers does not appear round in model

Chamfer accuracy for polybeams and contour plates

XS_CHAMFER_DISPLAY_LENGTH_FACTOR

Category: Model View

Use to adjust chamfer length. Tekla Structures calculates plate chamfer length from the plate side length using the following formula: `XS_CHAMFER_DISPLAY_LENGTH_FACTOR` * `plate side length`. By default this factor is `0.08`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`.

XS_CHANGE_DRAGGED_DIMENSIONS_TO_FIXED

Category

Drawing Properties

Use to define whether dimensions have fixed placing when you drag them away from their original locations. When set to `TRUE`, dimension placing uses fixed placing routine instead of free placing. `FALSE` keeps the placing free.

The default value is `TRUE`.

Note that this advanced option does not affect dimension type **Curved dimension, radial reference lines**.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_CHANGE_DRAGGED_MARKS_TO_FIXED

Category

Drawing Properties

Use to define whether part, bolt and reinforcement marks have fixed placing when you drag them away from their original locations. When set to `TRUE`, mark placing uses fixed placing routine instead of free placing.

The default value is `TRUE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_CHANGE_DRAGGED_NOTES_TO_FIXED

Category

Drawing Properties

Use to define whether associative notes have fixed placing when you drag them away from their original locations. When set to `TRUE`, associative notes placing uses fixed placing routine instead of free placing. `FALSE` keeps the placing free.

The default value is TRUE.

This advanced option is user specific and the setting is saved in options.bin under the user folder, for example, in C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings. Restart Tekla Structures to activate the new value.

XS_CHANGE_DRAGGED_TEXTS_TO_FIXED

Category

Drawing Properties

Use to define whether texts have fixed placing when you drag them away from their original locations. When set to TRUE, text uses fixed placing routine instead of free placing. The default value is TRUE.

This advanced option is user specific and the setting is saved in options.bin under the user folder, for example, in C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings. Restart Tekla Structures to activate the new value.

XS_CHANGE_DRAGGED_VIEWS_TO_FIXED

Category

Drawing Properties

Use to define whether views have fixed placing when you drag them away from their original locations. When set to TRUE, view placing uses fixed placing routine instead of free placing. If you set this advanced option to FALSE, fixed placing is not used. The default value is TRUE.

This advanced option is user specific and the setting is saved in options.bin under the user folder, for example, in C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings. Restart Tekla Structures to activate the new value.

XS_CHANGE_MARK_ASTERISK_TO

Category

Marking: parts

Use to specify a symbol to replace the asterisk character (*) in bolt marks and the size elements of part marks. The default value is asterisk (*).

This advanced option is model specific and the setting is saved in the options database.

NOTE You cannot use this advanced option to change the asterisk character in profile strings (e.g. PL10*100).

Example

```
XS_CHANGE_MARK_ASTERISK_TO=X
```

XS_CHANGE_WORKAREA_WHEN_MODIFYING_VIEW_DEPTH

Category

Model view

Makes working with reference files easier. Objects such as parts and reference files outside the work area are not visible, even when they are within the view depth range.

Set to `TRUE` (default) to automatically adjust the work area when you change the view depth.

XS_CHECK_BOLT_EDGE_DISTANCE_ALWAYS

Category

Modeling Properties

Use this advanced option to select whether to check the bolt edge distance. Set to `TRUE` to always check the bolt edge distance and to `FALSE` to check it only when there are bolts in the object (the check is not done if there is only a hole).

The default value is `TRUE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_CHECK_FLAT_LENGTH_ALSO

Category

Plate Work

Set this advanced option to `TRUE` (default) to check plate length and plate width, and then compare those with the possible dimensions for flat bars in `fltprops.inp`.

If you set this advanced option to `FALSE`, Tekla Structures only checks the plate width.

This advanced option is model specific and the setting is saved in the options database.

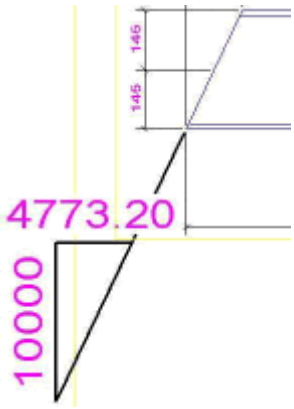
See also

XS_CHECK_TRIANGLE_TEXT_SIZE

Category

Dimensioning: General

Set this advanced option to `TRUE` to size the triangle to fit text in bevel dimensions. The default value is `FALSE`.



This advanced option is model specific and the setting is saved in the options database.

To define the angle symbol size, use the advanced option `XS_ANGLE_DIMENSION_SYMBOL_SIZE_FACTOR`.

NOTE Using advanced option `XS_ANGLE_DIMENSION_SYMBOL_SIZE_FACTOR` will disable advanced option `XS_CHECK_TRIANGLE_TEXT_SIZE`.

See also

XS_ANGLE_DIMENSION_SYMBOL_SIZE_FACTOR (page 79)

XS_CHORD_TOLERANCE_FOR_SMALL_TUBE_SEGMENTS

Category

Speed and Accuracy

Use this advanced option to define the chord tolerance for round tubes smaller than or equal to the limit set by the advanced option `XS_CHORD_TOLERANCE_SMALL_TUBE_SIZE_LIMIT`. Enter the value in millimeters. The default value is `1.0`.

Re-open the model to activate the new value.

NOTE Do not change the chord tolerance settings during a project. Changing them automatically recreates the tubular sections in the model when you restart Tekla Structures, resulting in slightly different solid objects, which may affect numbering.

See also

XS_CHORD_TOLERANCE_SMALL_TUBE_SIZE_LIMIT (page 120)

XS_CHORD_TOLERANCE_FOR_TUBE_SEGMENTS (page 119)

XS_CHORD_TOLERANCE_FOR_TUBE_SEGMENTS

Category

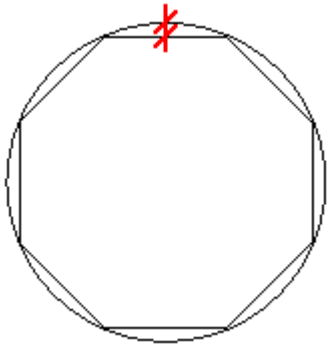
Speed and Accuracy

Use this advanced option to define the chord tolerance for round tubes larger than the limit set by the advanced option `XS_CHORD_TOLERANCE_SMALL_TUBE_SIZE_LIMIT`. Enter the value in millimeters. The default value is `1.0`.

Re-open the model to activate the new value.

Chord tolerance

Chord tolerance is the maximum difference between a straight segment used to display a tubular section in model views and the actual tubular section:



NOTE Do not change the chord tolerance settings during a project. Changing them automatically recreates the tubular sections in the model when you restart Tekla Structures, resulting in slightly different solid objects, which may affect numbering.

See also

XS_CHORD_TOLERANCE_SMALL_TUBE_SIZE_LIMIT (page 120)

XS_CHORD_TOLERANCE_FOR_SMALL_TUBE_SEGMENTS (page 118)

XS_CHORD_TOLERANCE_SMALL_TUBE_SIZE_LIMIT

Category

Speed and Accuracy

Use this advanced option to define the limit that determines whether a tube is considered small when calculating the chord tolerance. The value is compared to the diameter of the profile. Enter the value in millimeters. The default value is `50.0`.

Re-open the model to activate the new value.

NOTE Do not change the chord tolerance settings during a project. Changing them automatically recreates the tubular sections in the model when you restart Tekla Structures, resulting in slightly different solid objects, which may affect numbering.

See also

XS_CHORD_TOLERANCE_FOR_TUBE_SEGMENTS (page 119)

XS_CHORD_TOLERANCE_FOR_SMALL_TUBE_SEGMENTS (page 118)

XS_CIS_DEP1_DATABASE_NAME

Category

Export

Use to define the name of the temporary database used in CIMsteel transfers.

This advanced option is system specific and is read from environment files. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

XS_CIS_DEP1_DATABASE_PASSW

Category

Export

Use to define the password for the temporary database used in CIMsteel transfers.

This advanced option is system specific and is read from environment files. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

XS_CIS_DEP1_DATABASE_PATH

Category

Export

Use to define a path to the temporary database that Tekla Structures creates while converting a model from/to CIMsteel STEP format.

This advanced option is system specific and is read from environment files. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

XS_CIS_DEP1_EXPRESS_FILE

Category

This advanced option must be set in an initialization (.ini) file.

Use this advanced option to specify the name of the `EXPRESS` file located in the system folder. The `EXPRESS` file describes the CIMsteel DEP1 logical

product model. It is used when you convert a Tekla Structures model to/from CIMsteel DEP1 STEP format.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_CLASH_CHECK_BETWEEN_PARTS

Category: Modeling Properties

Set to `FALSE` to detect clashes only between Tekla Structures objects and reference model objects and to `TRUE` to detect clashes also between Tekla Structures objects. The default is `TRUE`.

You can also use the **Between parts** setting in the **Clash Check Manager** dialog box to determine what is included in the clash check. When you open a new model or restart Tekla Structures, the setting in the **Clash Check Manager** dialog box is reset to match the value of this advanced option. Changing the setting in **Clash Check Manager** does not change the value of this advanced option.

See also

XS_CLASH_CHECK_BETWEEN_REFERENCES (page 122)

XS_CLASH_CHECK_INSIDE_REFERENCE_MODELS (page 123)

XS_CLASH_CHECK_BETWEEN_REINFORCING_BARS (page 123)

XS_CLASH_CHECK_BETWEEN_REFERENCES

Category: Modeling Properties

Use to define whether or not Tekla Structures should run the clash check also between reference models.

Set to `TRUE` if you want to perform clash check also between two reference models. The default value is `FALSE`.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

You can also use the **Between reference models** setting in the **Clash Check Manager** dialog box to determine what is included in the clash check. When you open a new model or restart Tekla Structures, the setting in the **Clash Check Manager** dialog box is reset to match the value of this advanced

option. Changing the setting in **Clash Check Manager** does not change the value of this advanced option.

See also

XS_CLASH_CHECK_BETWEEN_PARTS (page 122)

XS_CLASH_CHECK_INSIDE_REFERENCE_MODELS (page 123)

XS_CLASH_CHECK_BETWEEN_REINFORCING_BARS (page 123)

XS_CLASH_CHECK_BETWEEN_REINFORCING_BARS

Category

Modeling Properties

Set this advanced option to `TRUE` to detect clashes between Tekla Structures reinforcing bars, and between reinforcing bars and other Tekla Structures objects (steel parts, bolts, embeds, and reference objects). Set to `FALSE` to detect clashes between reinforcing bars and other Tekla Structures objects except between reinforcing bars. The default is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_CLASH_CHECK_BETWEEN_PARTS (page 122)

XS_CLASH_CHECK_BETWEEN_REFERENCES (page 122)

XS_CLASH_CHECK_INSIDE_REFERENCE_MODELS (page 123)

XS_CLASH_CHECK_INSIDE_REFERENCE_MODELS

Category: Modeling Properties

Use this advanced option for defining whether internal clashes are checked in reference models.

The default value `FALSE` means that clashes between reference model objects within one reference model are ignored. Clashes between reference model objects belonging to different models are detected. The value `TRUE` means that clashes between reference model objects within one reference model are checked.

Note that this advanced option is taken into account only if the XS_CLASH_CHECK_BETWEEN_REFERENCES advanced option is set to TRUE.

You can also use the **Objects in reference models** setting in the **Clash Check Manager** dialog box to determine what is included in the clash check. When you open a new model or restart Tekla Structures, the setting in the **Clash Check Manager** dialog box is reset to match the value of this advanced option. Changing the setting in **Clash Check Manager** does not change the value of this advanced option.

See also

XS_CLASH_CHECK_BETWEEN_REFERENCES (page 122)

XS_CLEAR_MODEL_HISTORY

Category

Speed and Accuracy

If you use model history log files, set this advanced option to TRUE to clear history data from the log file each time you open and save the model. The default value is FALSE.

This advanced option is system specific and is read from environment files. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

See also

XS_COLLECT_MODEL_HISTORY (page 126)

XS_CLONING_TEMPLATE_DIRECTORY

Category: File Locations

Enter a path to the folder that contains cloning template models used by the **Master Drawing Catalog**. You can have several paths separated by semicolon. This advanced option is set to %XSDATADIR%\environments\common\cloning_templates by default.

This advanced option is user specific and the setting is saved in options.bin under the user folder, for example, in C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings. Restart Tekla Structures to activate the new value.

See also

XS_CLOUD_SHARING_PROXY

NOTE We recommend you to set this advanced option in **File menu** --> **Sharing** --> **Sharing settings** --> **Tekla Model Sharing cache**.

Use this advanced option to set up a cache server that is used with the Tekla Model Sharing sharing service.

If needed, the advanced option can be set in initialization files. Set `XS_CLOUD_SHARING_PROXY` to `name of the server;port` in initialization files.

This advanced option is user-specific.

XS_CNC_CUT_PLANE_HEIGHT

Category

CNC

Set this advanced option to a value between `0.3` and `1.0` if you get the following warning in the `dstv_nc.log` file for some NC files:

```
Error: Can't find intersection between solid and nc plane.
Please try to adjust XS_CNC_CUT_PLANE_HEIGHT (0.3 .. 1.0)
```

You may get these warnings when you try to create NC files for folded parts. These NC files are not correct. For example, the external contours of the part may be entirely missing from the NC file. You will notice this, if you view a DXF file created from the NC file.

To use this advanced option, follow these steps:

1. Set this advanced option using a value bigger than `0.3`. Start with smaller values first. The internal default value is `0.3`.
2. Create the NC files for the problematic part.
3. Check the file `dstv_nc.log`, and if there is still warning for this part, repeat steps 1 to 4 with another value.
4. After creating the NC files without any error messages in the `dstv_nc.log`, remove the advanced option from the `user.ini` file as this advanced option may cause problems for some other parts.

This advanced option is model specific and the setting is saved in the options database.

XS_CNC_HOLE_DIAMETER_ROUNDING

Category

CNC

Defines the rounding factor of holes for NC files. NC files are accurate to 0.01 mm.

Use to round bolt diameters with very small differences in diameter to the same value. You may need to use different diameters because bolts can have different tolerance values in a Tekla Structures model. You can input more decimals into bolt tolerance values in the component dialog boxes than in the general bolt dialog box.

The rounding routine is used only when you create NC files.

Set the rounding factor the same value (in mm) as the drill tolerance of the machine tool. The default value is `0.00001`.

Tekla Structures divides the original hole diameter by the rounding factor, then rounds the result to the nearest integer, then multiplies it by the rounding factor. As a result, bolt diameters with very small differences in diameter are now rounded to the same value.

This advanced option is model specific and the setting is saved in the options database.

Example

Here the rounding factor is 1.5875 (1/16 inch), which results in the following:

- Diameter 26.99 mm (1” 1/16 from component dialog box) => 26.99/1.5875 => 17.00 => 17 => 17*1.5875 => 26.99 mm
- Diameter 27.00 mm (1” 1/16 from bolt dialog box) => 27.00/1.5875 => 17.01 => 17 => 17*1.5875 => 26.99 mm

XS_COLLECT_MODEL_HISTORY

Category

Speed and Accuracy

Tekla Structures is able to collect model history data on various objects, for example, parts in the inquire object dialog, reinforcing bars, and components in reports, etc. Set this advanced option to `TRUE` (default) to collect the data. If you do not want to collect the model history, set it to `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_CLEAR_MODEL_HISTORY (page 124)

XS_COMBINED_BOLT_DIM_CHARACTER

Category

Dimensioning: Bolts

Use to set the character in bolt dimensions. The default value is asterisk (*).

This advanced option is model specific and the setting is saved in the options database.

XS_COMPANY_SETTINGS_DIRECTORY

Category

This advanced option must be set in an initialization (.ini) file.

Use this advanced option to define a folder where the initialization file `company.ini` is located. This file contains enterprise-level settings. You can define the folder, for example, in an `options.ini` file located under the model, firm or project folder.

XS_COMPLEX_PART_MEMBERS_DO_NOT_HAVE_TO_BE_MAIN_PARTS

Category

Dimensioning: Parts

Use in complex part/ assembly dimensioning to have Tekla Structures dimension other than main parts as one. The default value is `FALSE`.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_COMPONENT_CATALOG_ALLOW_SYSTEM_EDIT

Category: Modeling Properties

Set this advanced option to `TRUE` to be able to edit the **Applications & components** catalog definition files that are located in the `XS_SYSTEM` folders.

Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

XS_COMPONENT_CATALOG_DO_REPORT_LEGACY_FILE_ISSUES

Category

This advanced option must be set in an initialization (.ini) file.

This advanced option is system specific and is read from `teklastructures.ini`. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

Set the advanced option to `TRUE` to enable the reporting of legacy component catalog issues to the message log in the **Applications and Components** catalog.

Restart Tekla Structures after changing the value to activate the new setting.

XS_COMPONENT_CATALOG_COMPACT_THUMBNAIL_SIZE

Category: Modeling Properties

Use to adjust the size of thumbnails to between 16 and 96 pixels in the compact view. The default is 40.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_COMPONENT_CATALOG_THUMBNAIL_SIZE (page 128)

XS_COMPONENT_CATALOG_THUMBNAIL_SIZE

Category: Modeling Properties

Use to adjust the size of thumbnails to between 16 and 96 pixels in the normal view. The default is 96 pixels.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_COMPONENT_CATALOG_COMPACT_THUMBNAIL_SIZE (page 128)

XS_CONCRETE_PART_NUMBERING_PREFIX

Category

Numbering

Use to define a numbering prefix for concrete parts. The default value is `Concrete`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_CONCRETE_PART_NUMBERING_START_NUMBER (page 129)

XS_CONCRETE_PART_NUMBERING_START_NUMBER

Category

Numbering

Use to define a start number for concrete parts. The default value is 1.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_CONCRETE_PART_NUMBERING_PREFIX (page 129)

XS_CONNECTING_SIDE_MARK_SYMBOL

Category

Marking: Parts

Use to change the symbol for a connecting side mark in single-part, cast unit, and assembly drawings. By default the side mark symbol is number 34 in the `xsteel.sym` symbol file. To change the symbol, set this advanced option to a different symbol number.

In general arrangement drawings, use XS_GA_CONNECTING_SIDE_MARK_SYMBOL (page 263).

This advanced option is model specific and the setting is saved in the options database.

XS_CONNECT_CONNECTION_PARTS_IN_AUTOCONNECTION

Category: Components

Set this advanced option to `FALSE` (default) if you do not want to connect the parts created by the connection in Autoconnection. If you set it to `TRUE`, the parts created by the connection are connected in Autoconnection.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_CONNECT_PLATE_PROFILES_IN_AUTOCONNECTION

Category

Components

Set to `TRUE` to enable **AutoConnection** for built-up members. If you set this advanced option to `FALSE`, Tekla Structures does not consider plates as members when you use AutoConnection. The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

XS_CONNECT_UPLOAD_MODEL_FOLDER

Category

File Locations

Use this advanced option to define a Trimble Connect project folder where the current Tekla Structures model is uploaded as a read-only `.tekla` reference model. The `.tekla` reference model works as the counterpart of the Tekla Structures model in the linked Trimble Connect project. Uploading the Tekla Structures model as a `.tekla` reference model allows you to use ToDos and other Trimble Connect commands with the model.

Enter a path to the folder in the attached Trimble Connect project into which you want upload the current Tekla Structures model. By default, the folder path is `Structural\Tekla models`.

If you are working with a model that has been shared with Tekla Model Sharing, use the (page 458) advanced option to define when the shared model will be uploaded to the Trimble Connect project folder.

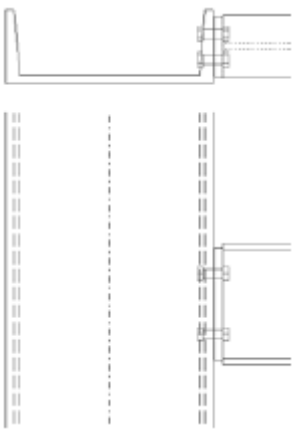
This advanced option is model specific and the setting is saved in the options database.

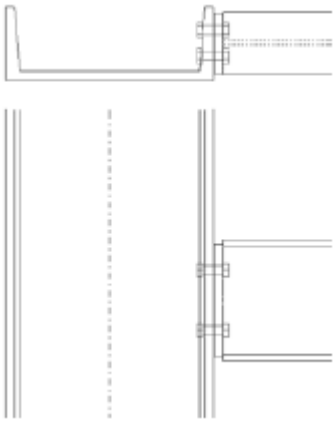
XS_CONSIDER_NEIGHBOUR_PARTS_IN_HIDDEN

Category

Drawing Properties

Set this advanced option to `TRUE` (default) to draw parts behind neighbor using hidden lines. If you want to show parts behind neighboring parts using a visible line type, set this advanced option to `FALSE`.

Setting	Example of appearance in drawing
Set to `TRUE` (default). Parts behind neighbor parts are drawn with hidden lines.	

Setting	Example of appearance in drawing
Set to FALSE. Parts behind neighbor parts are drawn with the visible line type.	

This advanced option is model specific and the setting is saved in the options database.

XS_CONSIDER_REBAR_HOOK_LOCATION_IN_CAST_UNIT_NUMBERING

This advanced option must be set in an initialization (.ini) file.

Use this advanced option to define whether the location of the reinforcing bar hooks are taken into account in cast unit numbering or not.

The default value is FALSE.

When this advanced option is set to TRUE, cast units that are otherwise identical will receive different numbers if the hooks in otherwise identical bars point to different directions, or if the stirrup hooks are in different corners, for example.

Restart Tekla Structures after changing the value to activate the new setting.

XS_CONSIDER_REBAR_NAME_IN_NUMBERING

Category

Concrete Detailing

Set to FALSE to have Tekla Structures only take part names into account when numbering, not reinforcing bar names. The default value is FALSE.

This advanced option is model specific and the setting is saved in the options database.

XS_CONTOUR_PLATE_POINT_ON_SAME_LINE_LIMIT

Category

Modeling Properties

Cleans away system component provided contour plate profile points in cases where points are on an almost straight line if the provided (middle) points deviate from the line at most as much as the value of this advanced option. The point is then so close to the line between the previous and next point that it is redundant and can be removed. Enter a decimal value. The default is 1.0 mm.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_CONTOUR_PLATE_POINT_ON_SAME_LINE_LIMIT_FOR_CLOSE_POINTS (page 133)

XS_CONTOUR_PLATE_POINT_ON_SAME_LINE_LIMIT_FOR_CLOSE_POINTS

Category

Modeling Properties

Cleans away system component provided contour plate profile points in cases where points are on an almost straight line if the provided (middle) points deviate from the line at most as much as the value of this advanced option. The point is then so close to the line between the previous and next point that it is redundant and can be removed. This advanced option is used if section point distances between two consecutive points are smaller than 10.0 mm. The default value for this advanced option is 0.1 mm.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_CONTOUR_PLATE_POINT_ON_SAME_LINE_LIMIT (page 133)

XS_CONVERSION_ARBITRARY_PROFILE_MAPPING_BY_NAME_MUST_MATCH_DIMENSIONS

Category: **Modeling Properties**

Set this advanced option to `TRUE` to require that the dimensions match the arbitrary profile mapping when mapping profile name to profile name in IFC object conversion. `FALSE` is the default value.

This advanced option is model specific and the setting is saved in the options database.

XS_CONVERT_OLD_FORCE_UNITS_TO_SI_FROM

Category

Components

Use to define how to convert part and connection attribute tables when you open a Tekla Structures catalog prior to version 7.0.

You can use the following units: kg/T/N/daN/kN/lbf/kip, or a numerical value.

This advanced option is model specific and the setting is saved in the options database.

Example

To convert force units from kip to SI, set this advanced option to `kip`.

XS_CONVERT_OLD_MOMENT_UNITS_TO_SI_FROM

Category

This advanced option must be set in an initialization (.ini) file.

Use to define how to convert part and connection attribute tables when you open a Tekla Structures catalog prior to version 7.0.

You can use the following units: kgm/Tm/Nm/daNm/kNm/lbf-in/lbf-ft/kip-in/kip-ft, or a numerical value.

Example

To convert moment units from kip-ft to SI, set this advanced option to `kip-ft`.

XS_COPY_REVISIONS_IN_AUTOMATIC_CLONING

Category

Drawing Properties

Set the advanced option `XS_COPY_REVISIONS_IN_AUTOMATIC_CLONING` to `TRUE` to copy the revisions and user-defined attributes to the automatically cloned drawings. `TRUE` is the default value.

This advanced option is model specific and the setting is saved in the options database.

XS_COUNT_ALL_PARTS_IN_NSFS_REPEATED_PART_MARK

Category

Marking: Parts

Use this advanced option to determine how the quantity of parts with merged part marks is shown. Set this advanced option to `TRUE` to count all parts. By default, this advanced option is set to `FALSE`.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

Example

When there are four or more identical stiffeners on both sides of a beam web, the mark is

- 4x1002BS if `XS_COUNT_ALL_PARTS_IN_NSFS_REPEATED_PART_MARK` is set to `TRUE`.
- 2x1002BS if `XS_COUNT_ALL_PARTS_IN_NSFS_REPEATED_PART_MARK` is set to `FALSE`.

See also

XS_COUNT_BOTH_PARTS_IN_NSFS_PART_MARK (page 135)

XS_COUNT_BOTH_PARTS_IN_NSFS_PART_MARK

Category

Marking: Parts

Use this advanced option to determine how the quantity of parts with merged part marks is shown. Set this advanced option to `TRUE` to count both parts. If you do not want to do this, set it to `FALSE` (default).

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

Example

When there are two identical stiffeners on both sides of a beam web, the mark is

- **2x1002BS** if `XS_COUNT_BOTH_PARTS_IN_NSFS_PART_MARK=TRUE`.
- **1002BS** if `XS_COUNT_BOTH_PARTS_IN_NSFS_PART_MARK=FALSE`.

See also

XS_COUNT_ALL_PARTS_IN_NSFS_REPEATED_PART_MARK (page 135)

XS_CREATE_ALSO_BIG_HTML_REPORT_PICTURES

Category

Concrete Detailing

Set to `TRUE` to create an additional set of pictures to the report folder. The pictures are three times larger in size compared to the ones in the HTML report. The default is `FALSE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_CREATE_DRAWING_PREVIEW_AUTOMATICALLY

Category

Drawing Properties

Set this advanced option to `TRUE` to take a snapshot of a drawing each time the drawing is saved. The snapshot is saved in the `\drawing` folder under the current model folder. The snapshot is used as the default preview image for

the drawing in the **Master Drawing Catalog**. If you set this advanced option to `FALSE`, the snapshot is not taken. The default value is `FALSE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_CREATE_MISSING_MARKS_IN_INTELLIGENT_CLONING

Category

Marking: General

Use to create all marks to a cloned drawing where new parts have been added.

To create the marks, set the advanced option to `TRUE`. The default is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

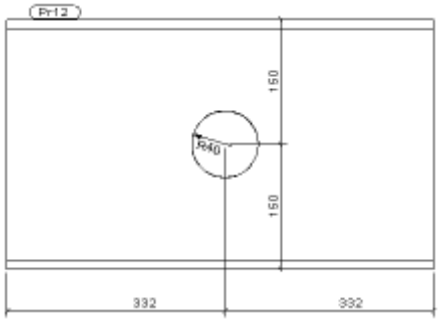
See also

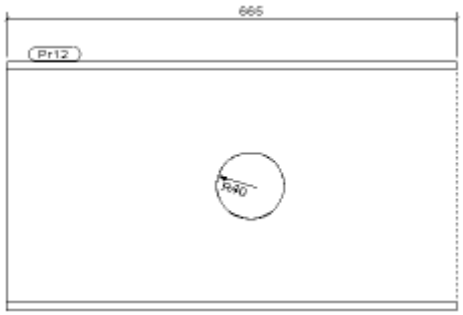
XS_CREATE_ROUND_HOLE_DIMENSIONS

Category

Dimensioning: Parts

Enter `TRUE` (default) to dimension the round hole center point in drawings.

Setting	Example of appearance in drawings
TRUE	

Setting	Example of appearance in drawings
FALSE	

You get the automatic dimension to hole center only if the cut part profile is round.

This advanced option does not apply to automatic general arrangement drawing dimensions.

This advanced option is model specific and the setting is saved in the options database.

XS_CREATE_CONNECTION_WHEN_COPYING_DRAWING_VIEWS

Category

Drawing Properties

Set to `TRUE` to link copied views to the original drawing. This means, for example, that when you delete a drawing from which you copied a view, Tekla Structures also deletes the copied view. The default value is `FALSE`.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

Useful links

For more information about using `XS_CREATE_CONNECTION_WHEN_COPYING_DRAWING_VIEWS`, see the following Support Article in Tekla User Assistance: What does the XS_CREATE_CONNECTION_WHEN_COPYING_DRAWING_VIEWS advanced option do?

XS_DRAWING_CREATE_SNAPSHOT_ON_DRAWING_CREATION

Category: Drawing Properties

Set this advanced option to `TRUE` to create a snapshot of a drawing at the same time that you create the drawing. If you set this option to `TRUE`, you no longer need to open a drawing to save it to create the snapshot. The default values is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_CREATE_VIEW_FROM_MODEL_OLD_WAY

Category

Drawing View

Use this advanced option to specify which values Tekla Structures uses for drawing view boundaries in general arrangement drawings. This only affects views created from the model.

If you set this advanced option to `FALSE` (default), the drawing view boundaries values are based on the current work area x and y coordinates of a model view.

When this advanced option is set to `TRUE`, the drawing view boundary values are based on the entire model view, ignoring the work area settings.

In both cases depth values are based on the model view display depth values.

This advanced option is model specific and the setting is saved in the options database.

XS_CS_CHAMFER_DIVIDE_ANGLE

Category: Speed and accuracy

Use this advanced option to change the angles dividing a cross section chamfer. The default value is `30.0` degrees.

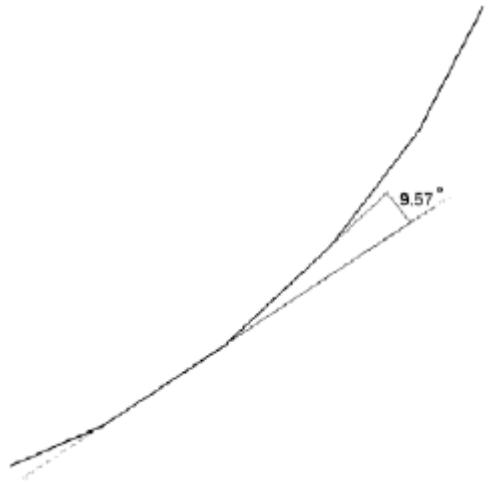
NOTE A very small value will cause profile roundings not to be visible.

Using this advanced option affects the rounding radius for parts that have high accuracy (for example parts in drawings and in DWG export).

This advanced option is model specific and the setting is saved in the options database.

Example

```
XS_CS_CHAMFER_DIVIDE_ANGLE=10.0
```



TIP When you want to export B-rep as exact solids in the IFC export, you need to set the advanced option XS_EXPORT_BREP_AS_EXACT_SOLID (page 252) to `TRUE`. To get smoother edges to the export, set `XS_CS_CHAMFER_DIVIDE_ANGLE` to `10`.

See also

XS_SOLID_USE_HIGHER_ACCURACY (page 438)

XS_CURVED_AXIS_PLACE

Category

Profiles

By default, Tekla Structures calculates the length of curved parts along the center axis. Use this advanced option to define the position of this axis as a ratio.

Tekla Structures calculates the location of the axis using the formula h=H/2.0*ratio. The default value is zero (0). Define the ratio as 1.0 to calculate length along the upper flange. Define the ratio as -1.0 to define the length along the lower flange.

This value is used in reports and drawing templates.

XS_CUSTOM_COMPONENT_DECIMALS

Category

Speed and Accuracy

Use this advanced option to adjust the number of decimals for custom components. This setting only works with parametric variables whose **Value type** is **Text**. Other parameter types follow the normal accuracy settings.

The default value is 3.

XS_CUT_SYMBOL_FONT

Category

Drawing Properties

Use to define the font for section symbol text. The default value is Arial. If you do not specify a font, Tekla Structures uses the default font defined for `XS_DEFAULT_FONT`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DEFAULT_FONT (page 144)

XS_CYCLIC_SOLVER_MAX_LOOPS

Category

Modeling properties

Use to define how many cycles Tekla Structures makes to solve the dependencies in custom components. Enter a number for the amount of cycle loops. The default is 2.

This advanced option is model specific and the setting is saved in the options database.

1.4 Advanced options - D

DAK_BMPPATH

This advanced option must be set in an initialization (.ini) file.

This advanced option is system specific and is read from `teklastructures.ini`. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

Use to point to the folder for bitmap files used in

- custom components
- user-defined attributes (`objects.inp` file)
- profile catalog
- some system components

You can use semicolon-separated lists of folder paths, see an example below. This advanced option is by default defined in the `teklastructures.ini` file and it points to the folder `..\ProgramData\Trimble\Tekla Structures\<version>\Bitmaps`

Example

```
DAK_BMPPATH=%XSDATADIR%\Bitmaps\
```

```
DAK_BMPPATH=%XSDATADIR%\Bitmaps\;H:\Tekla\bitmap\
```

XSDATADIR

Category

This advanced option must be set in an initialization (.ini) file.

This advanced option is system specific and is read from `teklastructures.ini`. It can also be set locally, see your environment `.ini` file (`env_<environment_name>.ini`). Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

`XSDATADIR` is defined in the `teklastructures.ini` file. It points to a location where the installation installs the environment files and folders.

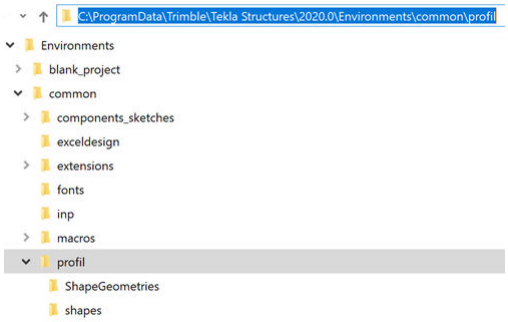
Example

By default, this advanced option is set as follows: `set XSDATADIR=C:\ProgramData\Trimble\Tekla Structures\<version>\`

XS_DEFAULT_BREP_PATH

This advanced option must be set in an initialization (.ini) file.

Use to define the location of the default shapes that are available in the shape catalog when you create a new model. By default, this advanced option is set to point to the `common` environment, and the definition files of the default shapes are read from the `\Shapes` and `\ShapeGeometries` folders under the `\profil` folder.



The definition files of the shapes that you use from this location when you create items in a model are copied to the model folder.

Example

To have Tekla Structures read the default shapes from the German environment, set this advanced option as follows:

```
set XS_DEFAULT_BREP_PATH=%XSDATADIR%\environments\germany\
```

or

```
set XS_DEFAULT_BREP_PATH=C:\ProgramData\Trimble\Tekla
Structures\<version>\environments\germany\
```

XS_DEFAULT_ENVIRONMENT

This advanced option must be set in an initialization (.ini) file.

Use together with XS_DEFAULT_LICENSE (page 145) and XS_DEFAULT_ROLE (page 147) for bypassing the login dialog. Set these advanced options in a separate initialization file, and point to that file using the -I (capital i) parameter in the Tekla Structures shortcut. The parameter is used to point to a file which needs to be read **before** other initialization files.

WARNING If roles are used in your environment, both `XS_DEFAULT_ENVIRONMENT` and `XS_DEFAULT_ROLE` should be set for the startup to work correctly. `XS_DEFAULT_LICENSE` is optional.

Example

```
set XS_DEFAULT_ENVIRONMENT=C:\ProgramData\Trimble\Tekla
Structures\<version>\Environments\germany\env_Germany.ini
```

XS_DEFAULT_FONT

Category: Model view

Use to specify the default font in the model and drawings, for example, when creating grids, dimensions, and mark texts. Enter the name of any Windows font, for example, Arial Narrow. The default value is `Segoe UI semibold`.

If any of the following advanced options are not set, or if font conversion is needed in drawings, Tekla Structures uses `XS_DEFAULT_FONT`:

- `XS_CUT_SYMBOL_FONT`
- `XS_DIMENSION_FONT`
- `XS_GRID_TEXT_FONT`
- `XS_MARK_FONT`
- `XS_VIEW_TITLE_FONT`
- `XS_WELD_FONT`

For example, if `XS_MARK_FONT` has no value set and you open an old model with a newer version of Tekla Structures, `XS_DEFAULT_FONT` is used when converting marks to the new model database.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_DEFAULT_FONT_SIZE (page 144)

XS_CUT_SYMBOL_FONT (page 141)

XS_DIMENSION_FONT (page 157)

XS_GRID_TEXT_FONT (page 268)

XS_MARK_FONT (page 304)

XS_VIEW_TITLE_FONT (page 498)

XS_WELD_FONT (page 501)

XS_DEFAULT_FONT_SIZE

Category

Model View

Use to set the default font size when modeling. The default value is `12`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_DEFAULT_HEIGHT_FOR_CALCULATED_DRAWING_SIZE

Category

Drawing Properties

Use to change the default height for the calculated drawing size. Enter the value in millimeters. The default value is `287`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DEFAULT_KEEP_ONLINE_LICENSE_CHECKBOX

This advanced option must be set in an initialization (.ini) file.

Use this advanced option for setting the default value of the **Keep this license reserved on this device** check box in the closing confirmation message box. The default value is `TRUE`, which keeps the license reserved. If this advanced option is not set, or is set to `TRUE`, the check box is selected if you use a named license, or if you have selected to keep the license reserved in the previous Tekla Structures session.

If you set this advanced option to `FALSE`, the **Keep this license reserved on this device** check box is never selected by default.

XS_DEFAULT_LICENSE

This advanced option must be set in an initialization (.ini) file.

Use to set the default license for a user role. The advanced option can be used either in a role-specific initialization file (`role_*.ini`) for setting the default license for a selected role, or in a separate initialization file together with XS_DEFAULT_ROLE (page 147) and XS_DEFAULT_ENVIRONMENT (page 143) for bypassing the login dialog.

WARNING If roles are used in your environment, both `XS_DEFAULT_ENVIRONMENT` and `XS_DEFAULT_ROLE` should be set for the startup to work correctly. `XS_DEFAULT_LICENSE` must be set if there are more than one license type available.

Possible values are:

- `CARBON`
- `GRAPHITE`
- `DIAMOND`
- `FULL`
- `TeklaStructures_Primary`
- `STEEL_DETAILING`
- `REBAR_DETAILING`
- `PRECAST_CONCRETE_DETAILING`
- `CONSTRUCTION_MODELING`
- `DRAFTER`
- `ENGINEERING`
- `PROJECT_VIEWER`
- `PRODUCTION_PLANNER`
- `EDUCATIONAL`
- `DEVELOPER`

See also

XS_DEFAULT_ROLE (page 147)

XS_DEFAULT_ENVIRONMENT (page 143)

XS_DEFAULT_MODEL_TEMPLATE

Category

This advanced option must be set in an initialization (.ini) file.

Use to define the model template which is used as the default in **File menu** --> **New** when you create a new model.

Define a value for the advanced option in the role initialization files, available in your environment folders. Store the default model template to the folder defined by the advanced option XS_MODEL_TEMPLATE_DIRECTORY (page 322).

If this advanced option is not set, the model template that was used last is displayed in **File menu** --> **New**.

Example

```
set XS_DEFAULT_MODEL_TEMPLATE=EngineeringTemplate
```

XS_DEFAULT_ROLE

This advanced option must be set in an initialization (.ini) file.

Use together with XS_DEFAULT_LICENSE (page 145) and XS_DEFAULT_ENVIRONMENT (page 143) for bypassing the login dialog. Set these advanced options in a separate initialization file, and point to that file using the -I (capital i) parameter in the Tekla Structures shortcut. The parameter is used to point to a file which needs to be read **before** other initialization files.

WARNING If roles are used in your environment, both `XS_DEFAULT_ENVIRONMENT` and `XS_DEFAULT_ROLE` should be set for the startup to work correctly. `XS_DEFAULT_LICENSE` is optional.

Example

```
set XS_DEFAULT_ROLE=C:\ProgramData\Trimble\Tekla Structures
\<version>\Environments\USA\Role_Imperial_Steel_Detailing.ini
```

XS_DEFAULT_WIDTH_FOR_CALCULATED_DRAWING_SIZE

Category

Drawing Properties

Use to change the default width for the calculated drawing size. Enter the value in millimeters. The default is `410`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DELETE_UNNECESSARY_DG_FILES

Category

Drawing Properties

Set the advanced option to `TRUE` (default) to delete unnecessary drawing files. If you do not want to do this, set it to `FALSE`. This advanced option deletes the dg files older than seven days, which is the default value for the advanced option `XS_DELETE_UNNECESSARY_DG_FILES_SAFETY_PERIOD`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DELETE_UNNECESSARY_DG_FILES_SAFETY_PERIOD (page 148)

XS_DELETE_UNNECESSARY_DG_FILES_SAFETY_PERIOD

Category

Drawing Properties

Defines the time frame after which unnecessary drawing files are deleted. Enter the required time frame in days. The advanced option is set to `7` by default.

WARNING We recommend that you do not modify the value of this advanced option unless you are an experienced Tekla Structures user.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DELETE_UNNECESSARY_DG_FILES (page 148)

XS_DELETE_UNNECESSARY_INT_ARRAYS

Category: Speed and Accuracy

Set to `TRUE` to also clean the int arrays when you go to the **File** menu and click **Diagnose and repair** --> **Repair model** . The default value is `FALSE`.

Create a backup copy of the model before you use this advanced option.

WARNING This setting reduces the model size in some cases but should be used with caution. if you have this advanced option enabled and run repair database, it deletes from the database all the integer and double arrays that are no longer used. If someone starts using integer arrays in a new location in the database, they would have to know about this functionality and update it so that those references would be found and no information would be lost.

XS_DELETE_UNNECESSARY_REFMODEL_FILES_SAFETY_PERIOD

Category : Multi-user

Defines the time frame after which unnecessary reference model files are deleted. Enter the required time frame in days. This advanced option is set to `7` by default. To use this advanced option, you need to set the advanced option`XS_REFERENCE_MODEL_KEEP_VERSIONS_COUNT (page 382)` to clean up the unnecessary reference models.

- In multiuser model, files are kept until safety period has passed.
- In shared model, files are kept until write out.

WARNING We recommend that you do not modify the value of this advanced option unless you are an experienced Tekla Structures user.

This advanced option has no effect on shared models.

This advanced option is model specific and the setting is saved in the options database.

XS_DETAIL_BOUNDARY_RADIUS

Category

Drawing Properties

Use to define a fixed size for circle-shaped detail boundaries in detail views.

Enter a numeric value in millimeters for the radius. By default, no size is set.

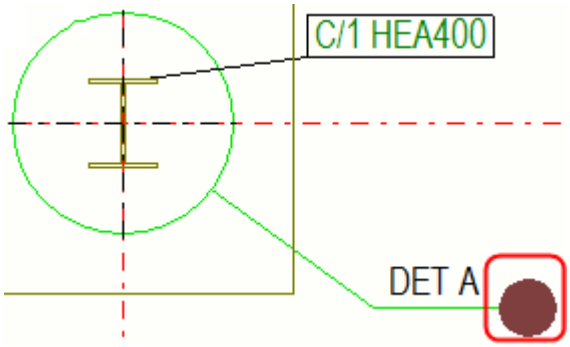
This advanced option is model specific and the setting is saved in the options database.

XS_DETAIL_MARK_REFERENCE_SYMBOL

Category

Drawing Properties

If you have set **Symbol** to **Custom** on the **Detail mark** tab page of the **Detail symbol properties** dialog box, Tekla Structures uses the value that you have set to this advanced option. For example, if you enter `xsteel@3`, Tekla Structures uses the symbol number 3 in symbol file xsteel.sym.



This advanced option is model specific and the setting is saved in the options database.

See also

XS_DETAIL_SYMBOL_REFERENCE

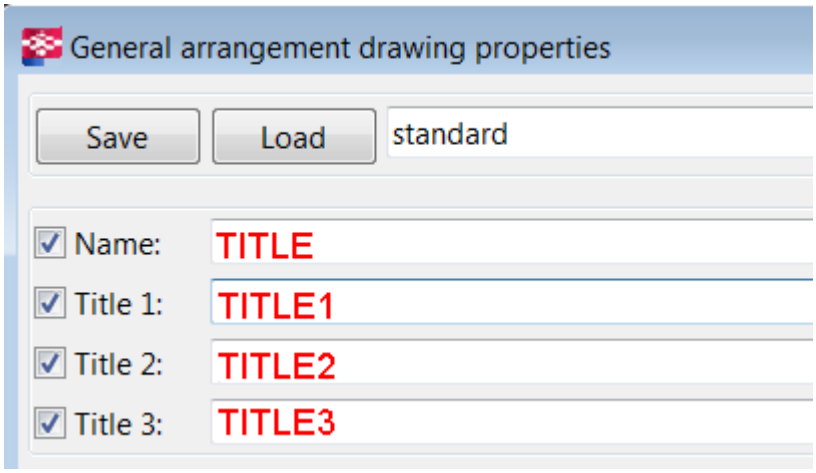
Category

Drawing Properties

Sets the reference text for symbols that show a detail from another drawing. The text can include:

- free text
- user-defined attributes
- template attributes

In the **Advanced options** dialog box, use single % characters around the user-defined and template attributes. `%DRAWING_TITLE%` is the default value. `%TITLE%` gives the same result. This advanced option gets the drawing name entered in the drawing properties dialog box. If you enter `TITLE1 - TITLE3`, Tekla Structures gets the drawing title from the drawing properties dialog box. You can also use the format `DR_TITLE1 - DR_TITLE3`.



This advanced option is model specific and the setting is saved in the options database.

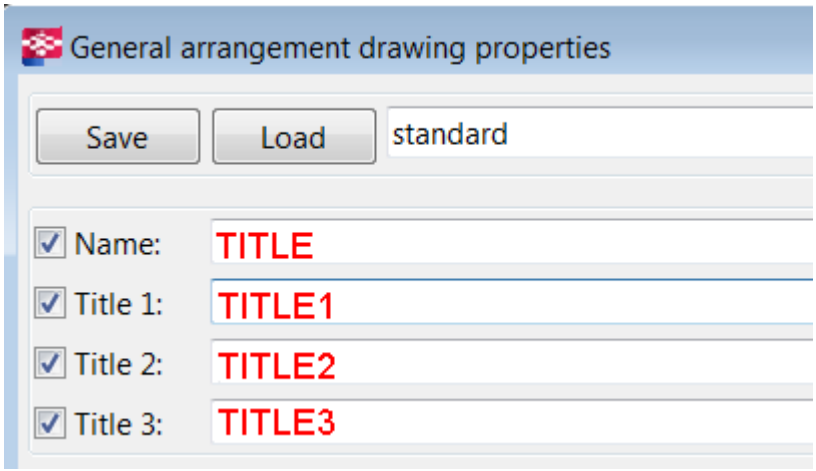
XS_DETAIL_VIEW_REFERENCE

Category

Drawing Properties

Set the reference text for detail view labels. You can enter free text, user-defined, attributes, and template attributes

Use single % characters around the user-defined and template attributes. `%DRAWING_TITLE%` is the default value. `%TITLE%` gives the same result. This advanced option gets the drawing name entered in the drawing properties dialog box. If you enter `TITLE1 - TITLE3`, Tekla Structures gets the drawing title from the drawing properties dialog box. You can also use the format `DR_TITLE1 - DR_TITLE3`.



This advanced option is model specific and the setting is saved in the options database.

See also

XS_DETAIL_SYMBOL_REFERENCE (page 150)

XS_DGN_EXPORT_PART_AS

Category

This advanced option must be set in an initialization (.ini) file.

Use to specify how Tekla Structures exports solid objects to DGN format.

- Set to `CELL` to export solid objects as cells containing the solid faces as a DGN shape.
- Set to `SOLID` to export solid objects as DGN solid objects defined by boundary elements.

The default value is `CELL`.

XS_DGN_EXPORT_USE_LOCAL_ID

Category

Export

Set this advanced option to `TRUE` to enable the use of local ID numbers in 3D DGN export. Every exported part gets a unique ID number starting from 1. The

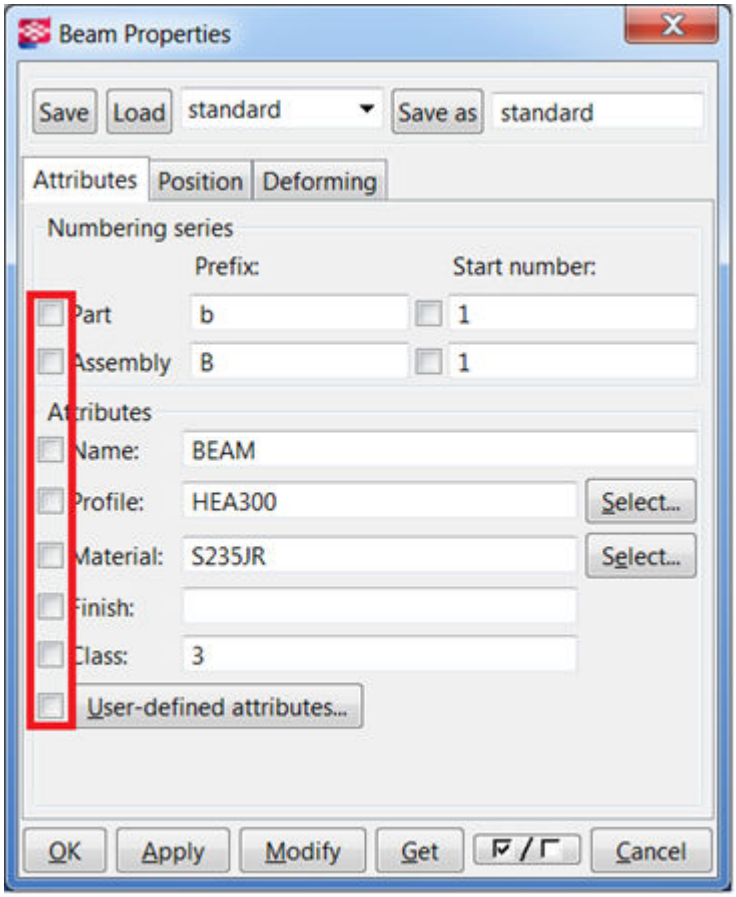
ID is saved as a user-defined attribute, and the same ID will be used in consequent exports. The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

XS_DIALOG_ENABLE_STATE

This advanced option must be set in an initialization (.ini) file.

Use this advanced option to remove selections from dialog box check boxes. `TRUE` is the default value.



XS_DIMENSION_ALL_BOLT_GROUPS_SEPARATELY

Category

Dimensioning: Bolts

Set this advanced option to `TRUE` to prevent Tekla Structures from combining bolt group dimensions. To combine bolt group dimensions, set it to `FALSE` (default).

This advanced option is model specific and the setting is saved in the options database.

XS_DIMENSION_DECIMAL_SEPARATOR

Category: **Dimensioning: General**

Use to define the character to be used as a decimal separator in dimensions and level marks in drawings, templates and reports. The default value is a full stop (.).

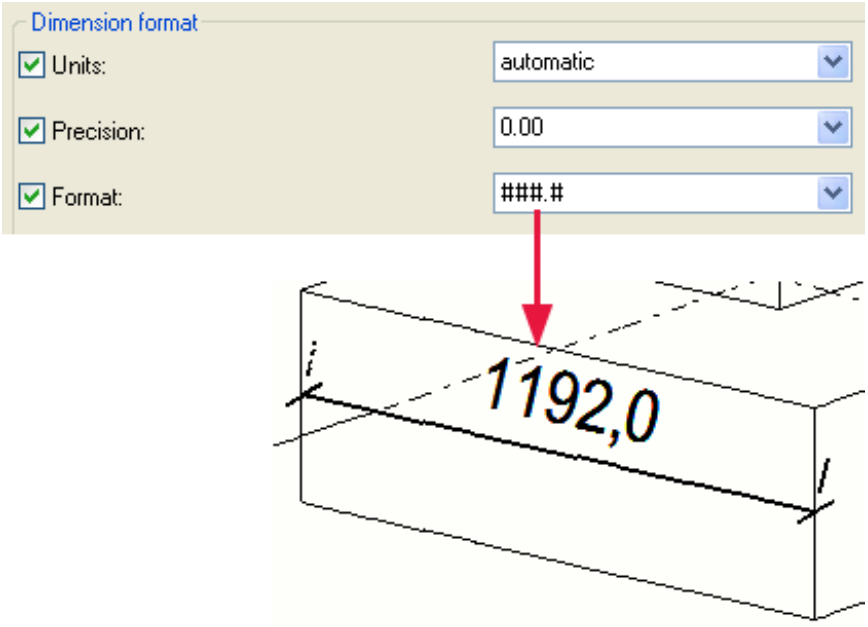
`XS_DIMENSION_DECIMAL_SEPARATOR = <separator character>`.

This advanced option is model specific and the setting is saved in the options database.

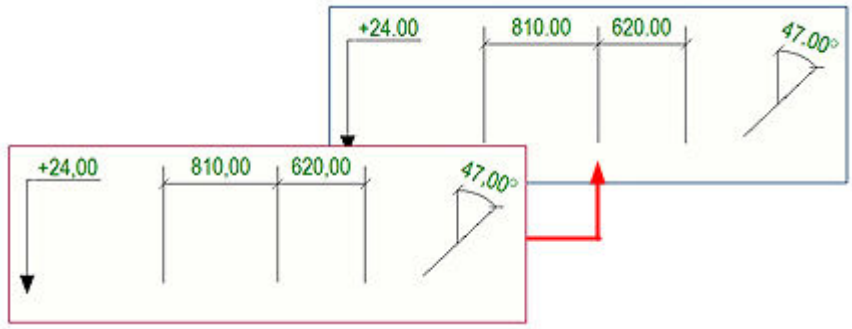
NOTE With this advanced option, you can change the separators in dimension objects in drawings, templates and reports, but not in part marks.

Example

In the following example, the dimension mark contains a comma when the dimension settings in the **Dimension Properties** dialog box are as follows, and the advanced option is set to `XS_DIMENSION_DECIMAL_SEPARATOR=,`



In the following example, the separator was first a period and was then changed to a comma.



XS_DIMENSION_DIGIT_GROUPING_CHARACTER

Category

Dimensioning: General

Use to specify which separator character to use in large dimension values. If you enter a comma (,), 154321 would become 154,321. If you do not specify a character, Tekla Structures uses a space in dimensions when you set grouping to**Yes** in **Dimension properties** > **General** > **Use grouping**. By default, no value is set.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DIMENSION_DIGIT_GROUPING_COUNT (page 155)

Dimension and dimensioning properties in drawings (page 684)

XS_DIMENSION_DIGIT_GROUPING_COUNT

Category

Dimensioning: general

Use to specify the number of digits after which a separator character will be inserted in dimension values. If you set this advanced option to 3 (default), 154321 becomes 154 321.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DIMENSION_DIGIT_GROUPING_CHARACTER (page 155)

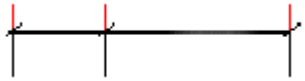
Dimension and dimensioning properties in drawings (page 684)

XS_DIMENSION_EXTENSION_LINE_AWAY_FACTOR

Category

Dimensioning: General

Use to adjust the length of the dimension extension lines that are facing away from the dimension points. Define the length as a factor for the dimension text size. The default is `1.0` (text height * 1.0).



This option is used only when the option **Short extension line** is set to **Yes** or **On grid lines only** in the **Dimension Properties** dialog box.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DIMENSION_EXTENSION_LINE_TOWARD_FACTOR (page 156)

XS_DIMENSION_EXTENSION_LINE_ORIGIN_OFFSET

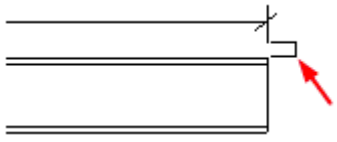
Category

Dimensioning: General

Use to specify the distance between the extension line origin and the start of the extension line (extension line origin offset). The default value is `1`.

This advanced option is model specific and the setting is saved in the options database.

In the following example, extension line origin offset is defined:

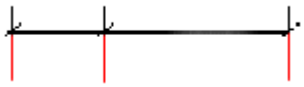


XS_DIMENSION_EXTENSION_LINE_TOWARD_FACTOR

Category

Dimensioning: General

Use to adjust the length of the dimension extension lines that are facing towards the dimension points. Define the length as a factor for the dimension text size. The default is `1.5` (text height * 1.5).



This option is used only when the option **Short extension line** is set to **Yes** or **On grid lines only** in the **Dimension Properties** dialog box.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DIMENSION_EXTENSION_LINE_AWAY_FACTOR (page 156)

XS_DIMENSION_FONT

Category

Drawing Properties

Use to specify the font for dimension text. The default value is Arial. If you do not specify a font, Tekla Structures uses the default font defined for `XS_DEFAULT_FONT` instead.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DEFAULT_FONT (page 144)

XS_DIMENSION_GROUPING_COUNT_SEPARATOR

Category

Dimensioning: General

Use to define the symbol that is displayed between the quantity and the automatic tag text in grouped dimensions. The default is `x`.

This advanced option is model specific and the setting is saved in the options database.

Note that the advanced option `XS_DIMENSION_GROUPING_COUNT_SEPARATOR` is intended for grouped dimensions that are defined in the integrated dimension settings (page 727). For other dimension marks, use the advanced option XS_DIMENSION_MARK_MULTIPLIER (page 159).

See also

XS_DIMENSION_LINE_TEXT_EPS

This advanced option has been removed, please use the advanced option XS_TEXT_ORIENTATION_EPSILON (page 449) instead.

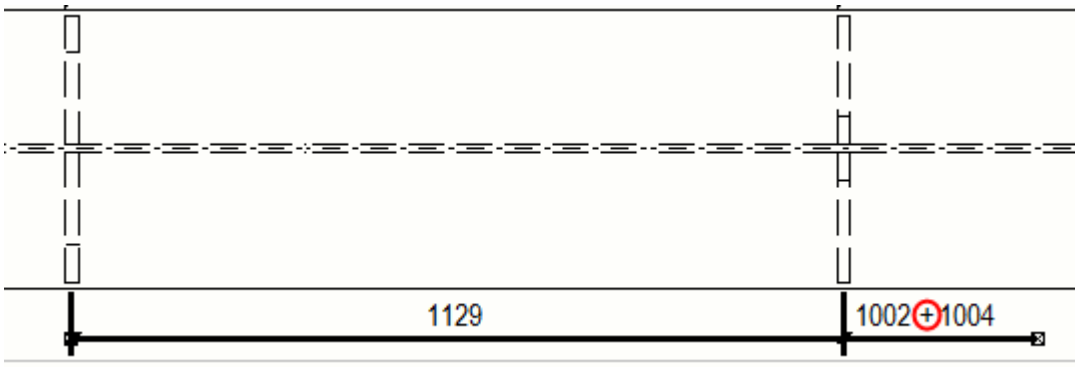
XS_DIMENSION_MARK_CONNECTOR

Category

Dimensioning: General

Use this advanced option to change the character that is used in dimension tags when the dimension tag displays the properties of the different objects dimensioned, for example, different part positions, and one dimension line is used. A plus sign is used by default.

This advanced option is model specific and the setting is saved in the options database.



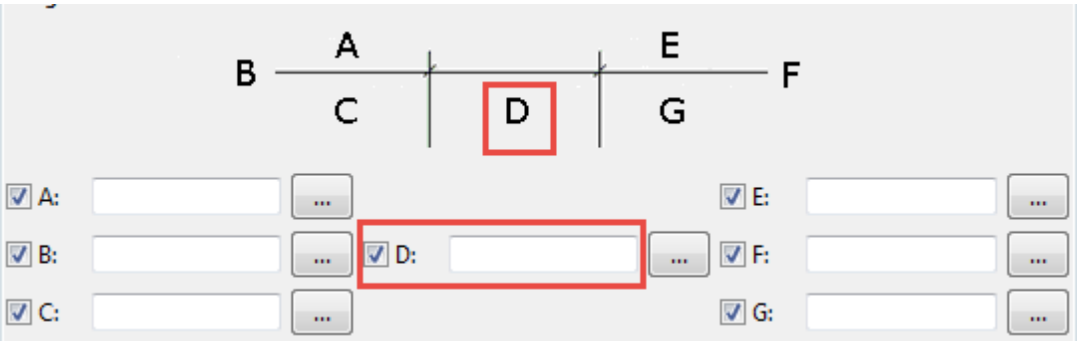
See also

Add tags to dimensions

XS_DIMENSION_MARK_CREATE_MIDDLE_TAG_ALWAYS

This advanced option must be set in an initialization (.ini) file.

Since Tekla Structures version 2017, the dimension middle tag functionality has been extended to show by default object information also when the dimension start and end are pointing to different objects. In previous versions, the middle tag content was only shown when the start and end were pointing to the same object. The previous functionality can be taken into use by setting this advanced option to `FALSE`.



XS_DIMENSION_MARK_MULTIPLIER

Category

Dimensioning: General

Use this advanced option to change the multiplication sign when there are several parts that have the same mark content, for example, HEA400 + 2 **x** HEA300. This advanced option has effect only if you have selected **Include part count in tag** in the **Dimension Properties** dialog box. `x` is the default value.

This advanced option is model specific and the setting is saved in the options database.

XS_DIMENSION_PART_MARK_CONTENT_IN_ASSEMBLY

Category

Dimensioning: Parts

Use for defining the content of the part mark placed next to the dimension line label in assembly drawings. Give any combination of the switches.

The available switches are:

- PROFILE
- MATERIAL
- SIZE
- LENGTH
- COMMENT
- WPDIST (distance between work points)
- GR_L (gross length of the part)

TIP If you want to display the part mark content in the same order as the switches, set XS_DIMENSION_PART_MARK_CONTENT_STRICT_POSITION to TRUE.

This advanced option is role-specific. When you change the value, it changes from system- to model-specific, and its value is the same for all users in the current model.

Example

```
XS_DIMENSION_PART_MARK_CONTENT_IN_ASSEMBLY=PROFILE_AND_LENGTH
.
XS_DIMENSION_PART_MARK_CONTENT_IN_ASSEMBLY=PROFILE_AND_GR_L
```

See also

XS_DIMENSION_PART_MARK_CONTENT_IN_SINGLE (page 160)

XS_DIMENSION_PART_MARK_CONTENT_STRICT_POSITION (page 161)

XS_DIMENSION_PART_MARK_CONTENT_IN_SINGLE

Category

Dimensioning: Parts

Use to define the content of the part mark that is placed next to the dimension line label in single-part drawings. Give any combination of the switches.

The available switches are:

- PROFILE
- MATERIAL
- SIZE
- LENGTH
- COMMENT

- WPDIST (distance between work points)
- GR_L (gross length of the part)

TIP If you want to display the part mark content in the same order as the switches, set XS_DIMENSION_PART_MARK_CONTENT_STRICT_POSITION to TRUE.

This advanced option is model specific and the setting is saved in the options database.

Example

```
XS_DIMENSION_PART_MARK_CONTENT_IN_SINGLE=PROFILE_AND_MATERIAL
XS_DIMENSION_PART_MARK_CONTENT_IN_SINGLE=PROFILE_AND_GR_L
```

See also

XS_DIMENSION_PART_MARK_CONTENT_IN_ASSEMBLY (page 159)

XS_DIMENSION_PART_MARK_CONTENT_STRICT_POSITION (page 161)

XS_DIMENSION_PART_MARK_CONTENT_STRICT_POSITION

Category

Dimensioning: Parts

Use for controlling the order of the switches in XS_DIMENSION_PART_MARK_CONTENT_IN_ASSEMBLY and XS_DIMENSION_PART_MARK_CONTENT_IN_SINGLE and thus the order of the part mark content in assembly and single drawings.

When set to TRUE, you can freely select the order of the switches in XS_DIMENSION_PART_MARK_CONTENT_IN_ASSEMBLY and XS_DIMENSION_PART_MARK_CONTENT_IN_SINGLE, and the part mark content follows the order of the switches. The default value is FALSE.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DIMENSION_PART_MARK_CONTENT_IN_ASSEMBLY (page 159)

XS_DIMENSION_PART_MARK_CONTENT_IN_SINGLE (page 160)

XS_DIMENSION_PLATE_SIDE_MARK_SYMBOL_CENTER

Category

Dimensioning: General

Use this advanced option to define the symbol Tekla Structures uses in center plate side marks. The default value is 1. Tekla Structures uses the symbols defined in `dimension_marks.sym` file located usually in the folder `..\Tekla Structures\<version>\environments\common\symbols\`.

This advanced option is model specific and the setting is saved in the options database.

XS_DIMENSION_PLATE_SIDE_MARK_SYMBOL_LEFT

Category

Dimensioning: General

Use this advanced option to define the symbol Tekla Structures uses in left plate side marks. The default value is 0. Tekla Structures uses the symbols defined in `dimension_marks.sym` file located usually in the folder `..\Tekla Structures\<version>\environments\common\symbols\`.

This advanced option is model specific and the setting is saved in the options database.

XS_DIMENSION_PLATE_SIDE_MARK_SYMBOL_RIGHT

Category

Dimensioning: General

Use this advanced option to define the symbol Tekla Structures uses in right plate side marks. The default value is 2. Tekla Structures uses the symbols defined in `dimension_marks.sym` file located usually in the folder `..\Tekla Structures\<version>\environments\common\symbols\`.

This advanced option is model specific and the setting is saved in the options database.

XS_DIMENSION_SKEWED_BOLTS_IN_PART_PLANE_IN_SINGLE_DRAWINGS

Category

Dimensioning: Bolts

Set this advanced option to `TRUE` to dimension bolts perpendicular to the part plane in single-part drawings. If you do not want to do this, set it to `FALSE`. The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

XS_DIR

This advanced option is deprecated. You can still use it in Tekla Structures version 2022 and older Tekla Structures versions. Instead of this advanced option, we recommend that you use XSDATADIR (page 142) to find the root location for the installation, or XSBIN (page 97) to find the location where binaries are found.

This advanced option must be set in an initialization (.ini) file.

This advanced option points to the folder for Tekla Structures data, source, and binary files.

This advanced option is system specific and is read from `teklastructures.ini`. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

XS_DISABLE_ADVANCED_OPTIONS

Category

This advanced option must be set in an initialization (.ini) file.

You can disable the **Advanced Options** dialog box so that the advanced options can be edited only in the initialization files.

To disable the dialog box, set this advanced option to `TRUE`. If you do not want to disable the dialog box, set it to `FALSE`. The default value is `FALSE`.

See also

XS_DISABLE_ANALYSIS_AND_DESIGN

Category

Analysis & Design

Set to `TRUE` to disable the following analysis and design tools from the Tekla Structures user interface:

- The following command from the **Analysis** menu and the related icons from the **Loads and Analysis** toolbar:
  - **Analysis & Design Models**
- The following tabs from the part properties dialog boxes:
  - **Analysis**
  - **Loading**
  - **Composite**
  - **Spanning**
  - **Start releases**
  - **End releases**
  - **Design**
- The two **Analysis** tabs from the parts' user-defined attributes dialog boxes

The default value is `FALSE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_DISABLE_CANCEL_DIALOG_FOR_SAVE_NUMBERING_SAVE

Category

Numbering

Use to revert to the old numbering functionality that offers no option to cancel numbering before the second save is made. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_DISABLE_CLASSIFIER_FOR_MODIFIED_PARTS

Category: Drawing Properties

Use to disable the Detailed object level settings check for modified parts. When you change part properties in a model after you have defined the Detailed object level settings, part presentation and mark content are updated in general arrangement drawings unless you have set this advanced option to `TRUE`.

To disable the check for modified parts, set the advanced option to `TRUE`.

To enable the check for modified parts, set the advanced option to `FALSE`. The default value is `FALSE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_DISABLE_DRAWING_PLOT_DATE

Category

Drawing Properties

Set this advanced option to `TRUE` to disable the print date information in drawings. Disabling the print date helps to avoid possible conflicts when working with multi-user models in situations where a user modifies drawings while another user prints the same drawings. Set to `FALSE` to have the print date information.

By default, the advanced option is set to `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

XS_DISABLE_PARTIAL_REFRESH

Category

Model View

Set this advanced option to `TRUE` to disable partial refreshing of OpenGL windows. This advanced option is for older ATI graphics cards. If you do not want to disable the partial refresh, set the advanced option to `FALSE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_DISABLE_REBAR_MODELING

Category

Analysis & Design

If you set this advanced option to `TRUE`, Tekla Structures removes reinforcing bar modeling module, even if you have a license for this module. This license is always included, except in the Viewer configuration. The default is `FALSE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_DISABLE_TEMPLATE_DOUBLE_CLICK

Category: **Drawing Properties**

By default, you are allowed to edit drawing templates and table layouts. It is possible to prevent the template and table layout editing by setting the advanced option `XS_DISABLE_TEMPLATE_DOUBLE_CLICK` to `TRUE`. When you do this:

- You cannot start editing templates or table layouts by double-clicking a template in a drawing. Instead, the **Drawing properties** dialog box will be displayed.
- You cannot start editing templates or table layout by right-clicking a template in a drawing and selecting the corresponding commands.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_DISABLE_VIEW_CENTERING_ASSEMBLY

Category

Drawing View

Use to left-align and/or top-align assembly drawing views. You can use values HOR (left-align), VER (top-align), TRUE (left-align and top-align), or FALSE (no alignment). Leaving out the value gives the same result as FALSE.

This advanced option is model specific and the setting is saved in the options database.

XS_DISABLE_VIEW_CENTERING_GA

Category

Drawing View

Use to left-align or top-align GA drawing views. You can use values HOR (left-align), VER (top-align), or both. For creating centered views, leave this advanced option empty

This advanced option is model specific and the setting is saved in the options database.

XS_DISABLE_VIEW_CENTERING_MULTI

Category

Drawing View

Use to left-align or top-align multidrawing views. You can use values HOR (left-align), VER (top-align), or both.

This advanced option is model specific and the setting is saved in the options database.

XS_DISABLE_VIEW_CENTERING_SINGLE

Category

Drawing View

Use to left-align or top-align single-part drawing views. You can use values HOR (left-align), VER (top-align), or both.

This advanced option is model specific and the setting is saved in the options database.

XS_DISPLAY_DIMENSIONS_WHEN_CREATING_OBJECTS

Category

Model View

Use for defining whether dimensions and dimension lines are displayed when model objects are created.

When set to `TRUE` (default), Tekla Structures displays the dimensions and dimension lines.

Dimensions and dimension lines are displayed when you create a new model object and pick the start point and the intermediate or end points for the object.

If you set this advanced option to `FALSE`, the dimensions are not displayed.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_DISPLAY_DIMENSIONS_WHEN_SELECTING_OBJECTS

Category: Model View

Use for defining whether dimensions and dimension lines are displayed when you select a column or a beam.

When this advanced option is set to `TRUE` (default), Tekla Structures displays the dimensions and dimension lines.

Dimensions and dimension lines are displayed when you select a single object, or when you select multiple objects by picking them. The dimensions are not displayed if a command is running, or if you select multiple objects with area selection.

When this advanced option is set to `FALSE`, the dimensions and dimension lines are not displayed.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

NOTE When direct modification is switched on, the standard object dimensions and dimension lines are always hidden, regardless of these settings. Only the direct modification dimensions are then shown. This makes it easier to know which dimensions can be edited.

See also

XS_DISPLAY_DIMENSIONS_WHEN_SELECTING_REBARS (page 169)

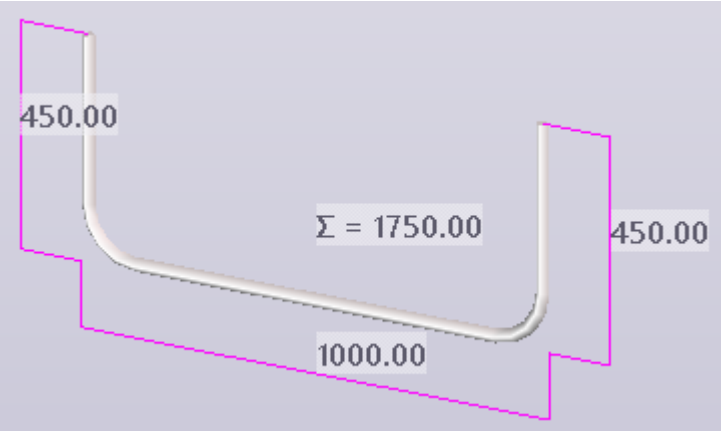
XS_DISPLAY_DIMENSIONS_WHEN_CREATING_OBJECTS (page 167)

XS_DISPLAY_DIMENSIONS_WHEN_SELECTING_REBARS

Category: Model View

Use for defining whether dimensions and dimension lines are displayed when you select a reinforcing bar, bar group, or rebar set.

When this advanced option is set to `TRUE` (default), Tekla Structures displays the leg dimensions, total bar length (Σ), and dimension lines. For bar groups, including rebar sets, the dimensions of the first bar and the last bar are displayed. If you select individual bars in rebar sets, the dimensions of all the selected rebar set bars are displayed. Dimensions are not displayed for curved or circular bars, or for hooks.



For single reinforcing bars and bar groups, these dimensions are only displayed when direct modification is switched off. When direct modification is switched on, these dimensions are always hidden, and the direct modification dimensions are shown instead. This makes it easier to know which dimensions can be edited.

Dimensions and dimension lines are displayed when you select a single object, or when you select multiple objects by picking them. The dimensions are not displayed if a command is running, or if you select multiple objects with area selection.

When this advanced option is set to `FALSE`, the dimensions and dimension lines are not displayed.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`.

TIP To quickly switch between the values `TRUE` and `FALSE`, go the **Rebar** tab on the ribbon and click **Visibility** --> **Rebar dimensions**, or use the keyboard shortcut **Alt+6**.

See also

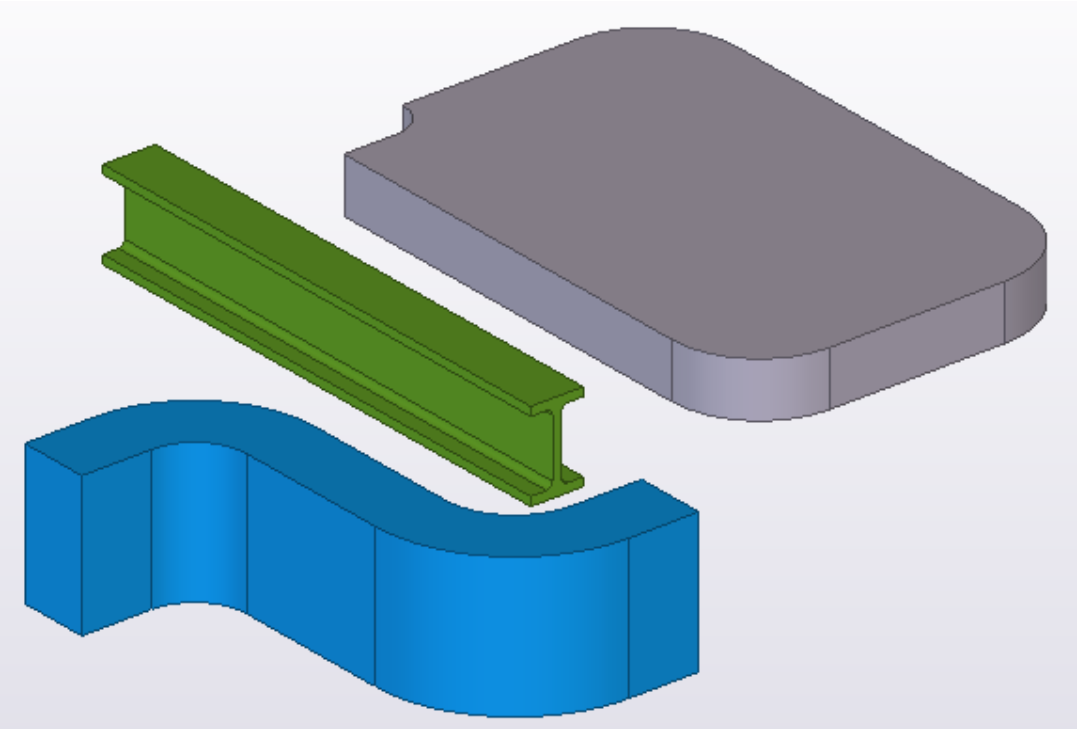
XS_DISPLAY_DIMENSIONS_WHEN_SELECTING_OBJECTS (page 168)

XS_DISPLAY_FILLET_EDGES

Category: Model View

Use this advanced option to show or hide fillet edges in model views. The default value is `TRUE`, which shows the fillet edges. If you do not want to show the fillet edges, set this advanced option to `FALSE`.

Fillet edges are lines that define the boundary between flat faces and curved faces in the model. Examples of model objects containing fillet edges are profiles with curved fillets, slabs or contour plates with curved chamfers, and curved polybeams.



The fillet edges in certain profiles are shown in model views only when you show parts with high accuracy.

This advanced option is user specific, and the setting is saved in `options.bin` under the user folder. Reopen the model to activate the new value.

See also

XS_SOLID_USE_HIGHER_ACCURACY (page 438)

XS_DISPLAY_ZERO_INCHES

Category: Imperial units

Set this advanced option to `TRUE` to display zero inches. The default value is `FALSE`.

Using this advanced option affects the results you get with the inquire tool, and the dimensions in the model and drawings.

Example:

`TRUE`: 2'-0"

`FALSE`: 2'

This advanced option is model specific and the setting is saved in the options database.

XS_DISTANT_OBJECT_FINDER_TOLERANCE

Category

Model View

Use to set the minimum distance from the nearest part for the reporting of distant objects. The object must be located outside this distance for it to be listed by the **Find Distant Objects** tool.

The distance is measured from the nearest part. Enter the value in meters. The default value is `100`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DO_NOT_CLIP_NATIVE_OBJECTS_WITH_CLIP_PLANE

Category: Model View

Set this advanced option to `TRUE` to only clip point clouds and reference models with the **Clip plane** command. Native Tekla Structures objects are not clipped. `FALSE` is the default value.

Redraw the model views after changing the value.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder.

XS_DO_NOT_CREATE_ASSEMBLY_DRAWINGS_FOR_CONCRETE_PARTS

Category

Drawing Properties

Use to control the creation of single-part and assembly drawings from concrete parts.

To enable the creation of drawings, set the advanced option to `FALSE`.

To disable the creation of drawings, set the advanced option to `TRUE` (default).

This advanced option is model specific and the setting is saved in the options database.

XS_DO_NOT_CREATE_ASSEMBLY_DRAWINGS_FOR_LOOSE_PARTS

Category

Drawing Properties

Use this advanced option for defining whether assembly drawings are created for assemblies that contain only one part.

When you set this advanced option to `TRUE`, Tekla Structures does not create assembly drawings for single-part assemblies. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_DO_NOT_CREATE_BOLT_MARKS_IN_ALL_INCLUDED_SINGLE_VIEWS

Category

Single Part View in Assembly Drawing

Use to prevent the creating of bolt marks in included single views.

Enter TRUE to exclude the bolt marks, FALSE to create the bolt marks. The default is FALSE.

This advanced option is user specific and the setting is saved in options.bin under the user folder, for example, in C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings. Restart Tekla Structures to activate the new value.

XS_DO_NOT_CREATE_PART_MARKS_IN_ALL_INCLUDED_SINGLE_VIEWS

Category

Single Part View in Assembly Drawing

Use to prevent the creating of part marks in included single part views.

Enter TRUE to exclude the part marks and FALSE to create the part marks. The default is FALSE.

This advanced option is user specific and the setting is saved in options.bin under the user folder, for example, in C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings. Restart Tekla Structures to activate the new value.

XS_DO_NOT_CREATE_PROFILE_DIMENSIONS_FOR_CONCRETE

Contents

Drawing Properties

Set this advanced option to TRUE to prevent Tekla Structures from automatically displaying the profile dimensions of concrete parts in cast unit drawings. If you want to display the profile dimensions, set it to FALSE.

This advanced option is model specific and the setting is saved in the options database.

XS_DO_NOT_DISPLAY_CHAMFERS

Category: Model View

Set this advanced option to TRUE to draw polygon plates without chamfer lines. If you want to display the chamfer lines, set this advanced option to FALSE. The default value is FALSE.

This advanced option works with XS_DRAW_CHAMFERS_HANDLES (page 190) set to CHAMFERS or to CHAMFERS_AND_HANDLES.

This advanced option is user specific and the setting is saved in options.bin under the user folder, for example, in C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings. Restart Tekla Structures to activate the new value.

XS_DO_NOT_DRAW_COLUMN_MARKS_AT_45_DEGREES_IN_GA_DRAWING

Category

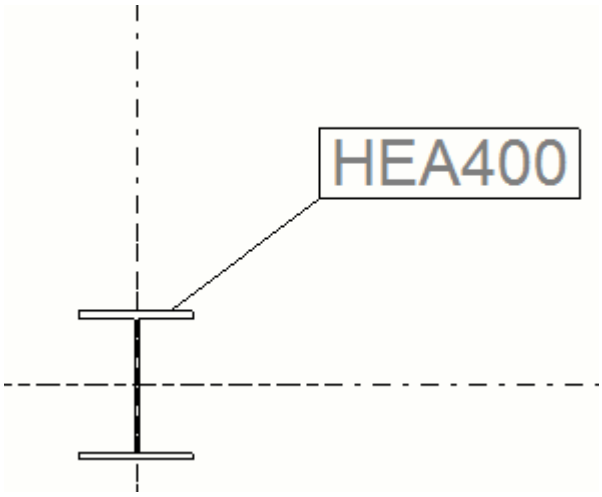
Marking: Parts

Tekla Structures places column mark texts in general arrangement drawing plan views by default at an angle of 45 degrees in respect to the position of the column. To place the marks horizontally, set this advanced option to TRUE. If you set this advanced option to FALSE, the mark texts are placed at the angle of 45 degrees. The default value is TRUE.

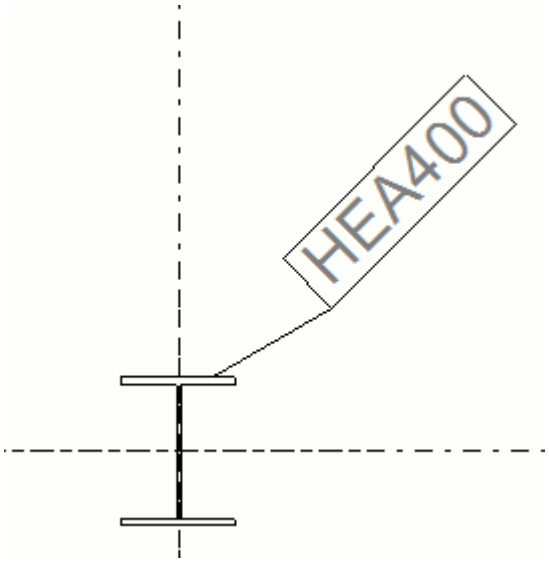
This advanced option is model specific and the setting is saved in the options database.

Example

In the following example, the advanced option is set to TRUE.



In the following example, the advanced option is set to FALSE.



XS_DO_NOT_EXTEND_DIMENSION_LINES_THROUGH_ALL_HOLES

Category

Dimensioning: Bolts

Set this advanced option to `TRUE` (default) to prevent dimension lines from extending through all holes in a bolt group. If you wish to extend the dimension lines, set this advanced option to `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_DO_NOT_PLOT_DIMENSION_POINT_CIRCLES

Category: Dimensioning: General

Set to `TRUE` to prevent Tekla Structures from printing red dimension point invalidity symbols when you print from **Document manager**. Dimension point invalidity symbols are always printed if you have a drawing open. The default value is `FALSE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_DO_NOT_OVERWRITE_PLUGIN_INP_FILE

Category

This advanced option must be set in an initialization (.ini) file.

With standard settings Tekla Structures overwrites the step profile settings and replaces them with the default values every time when Tekla Structures is started. To prevent Tekla Structures from losing the step profile settings, set the advanced option `XS_DO_NOT_OVERWRITE_PLUGIN_INP_FILE` to `TRUE` in the `teklastructures.ini` file.

If you are using catalog step profiles and have set `XS_DO_NOT_OVERWRITE_PLUGIN_INP_FILE` to `TRUE` and you update Tekla Structures, do the following:

1. Set `XS_DO_NOT_OVERWRITE_PLUGIN_INP_FILE` to `FALSE` in `teklastructures.ini` file.
2. Update Tekla Structures.
3. Start Tekla Structures.
4. Set `XS_DO_NOT_OVERWRITE_PLUGIN_INP_FILE` to `TRUE` in `teklastructures.ini` file.
5. Run `Steps.exe`.
6. Restart Tekla Structures.

See also

Ship Ladder (page 2321)

XS_DO_NOT_REMOVE_END_ABSOLUTE_DIMENSIONS

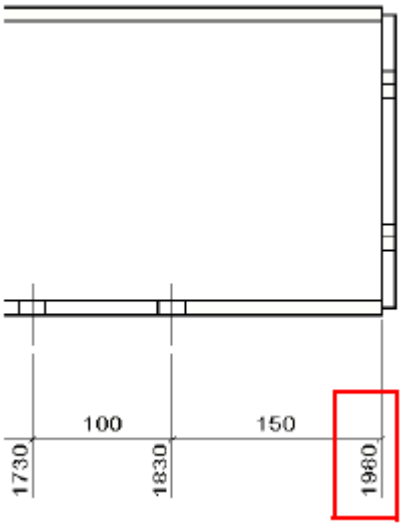
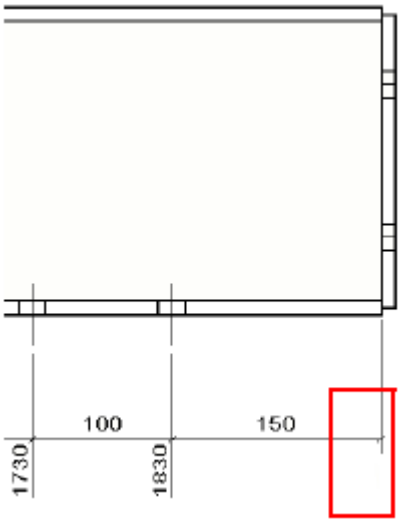
Category

Dimensioning: Parts

When you use absolute dimensions, Tekla Structures removes the last absolute vertical dimension. To prevent this, set this advanced option to `TRUE` (default). If you do not want to do this, set this advanced option to `FALSE`.

Note that XS_DO_NOT_REMOVE_END_ABSOLUTE_DIMENSIONS only works when the dimensioning type is**Integrated dimensions**.

This advanced option is model specific and the setting is saved in the options database.

Setting	Example of appearance in drawings
TRUE	
FALSE	

XS_DO_NOT_USE_FOLDED_GUSSET_PLATE

This advanced option must be set in an initialization (.ini) file.

Set this advanced option to TRUE to use part-adds to create bent plates in gusset connections. If you set it to FALSE, Tekla Structures creates bent gusset plates using the **Polybeam** command instead of the **Part-add** command.

XS_DO_NOT_USE_GLOBAL_PLATE_SIDE

Category

Components

By default, connections 141, 146, 147, 149, 181, 184, 185, 186 and 187 create all shear tabs on the same side of the main part if the main part has several connections (global positioning). Set this advanced option to `TRUE` to position each shear tab according to which end of the main part the connection that creates it is closest to (local positioning). The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

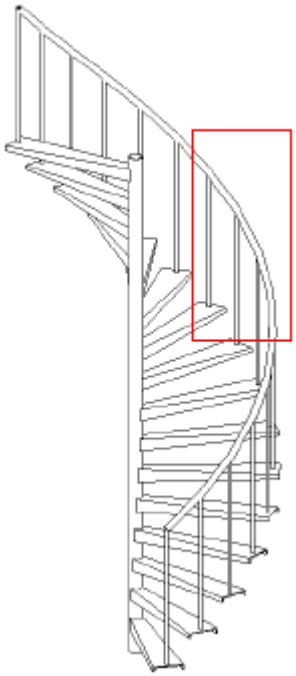
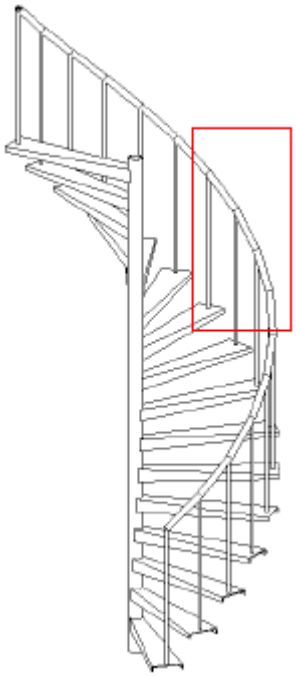
XS_DONT_SHOW_POLYBEAM_MID_EDGES

Category: Drawing Properties

Set this advanced option to `TRUE` to hide polybeam fold and bend lines in drawings. If you set it to `FALSE` (default), the fold and bend lines are shown.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`.

Example

Advanced option is set to	Example	Description
TRUE		Polybeam fold and bend lines are not shown in the handrail.
FALSE		Polybeam fold and bend lines are shown in the handrail.

XS_DRAW_ALL_SECTION_EDGES_IN_DRAWINGS

Category

Drawing Properties

Set this advanced option to `TRUE` to draw the edges of a cut part in the drawing and to `FALSE` to leave the edges open. The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SECTION_LINE_COLOR (page 400)

XS_DRAW_ANGLE_AND_RADIUS_INFO_IN_UNFOLDING

Category

Dimensioning: Unfolding

Set this advanced option to `TRUE` to show the angle and radius information on the dimension line. The default for this advanced option is that the information is shown. To hide this information, set it to `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DRAW_BENDING_END_LINE_DIMENSIONS_IN_UNFOLDING

Category: Dimensioning: Unfolding

Set this advanced option to `TRUE` to create bending end line dimensions when creating unfolded single-part drawings. The default value is `FALSE`.

Note that two different dimensions are created if the advanced option XS_DRAW_BENDING_LINE_DIMENSIONS_IN_UNFOLDING is set to TRUE as well.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DRAW_BENDING_END_LINES_IN_UNFOLDING (page 181)

XS_DRAW_BENDING_END_LINES_IN_UNFOLDING

Category: Dimensioning: Unfolding

Set this advanced option to TRUE to draw bending end lines when creating unfolded single part drawing of a bent plate or a polybeam. For polybeams, the lines will be drawn only if the polybeam has circular shape chamfers. The default value is FALSE.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DRAW_BENDING_END_LINE_DIMENSIONS_IN_UNFOLDING (page 180)

XS_DRAW_BENDING_LINE_DIMENSIONS_IN_UNFOLDING

Category

Dimensioning: Unfolding

Set this advanced option to TRUE to create bending line dimensions in a drawing.

If you do not want to create these dimensions, set this advanced option to FALSE. The default value is TRUE.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DRAW_BOLT_HIDDEN_LINES

Category: **Drawing properties**

Use the following advanced options to show or hide bolts that are hidden from view by other parts, in single part, assembly, and general arrangement drawings. The default values are shown here:

- `XS_DRAW_BOLT_HIDDEN_LINES_IN_SINGLE_DRAWINGS=AS_PART`
- `XS_DRAW_BOLT_HIDDEN_LINES_IN_ASSEMBLY_DRAWINGS=AS_PART`
- `XS_DRAW_BOLT_HIDDEN_LINES_IN_GA_DRAWINGS=FALSE`

Possible values are:

- `AS_PART`: uses part's hidden line visibility settings.
- `TRUE`: always shows hidden bolts.
- `FALSE`: does not show hidden bolts.

NOTE Tekla Structures sets the way the bolt hidden lines are shown when the bolt is added to a drawing, usually when you create the drawing or a drawing view. The hidden lines setting of the bolts cannot be changed afterwards.

One way to change the setting in an old drawing is to create a new drawing view using the desired bolt hidden lines settings.

Example

The following examples show how these advanced options can be used together with drawing property settings.

To	Do this
Set the hidden lines to be always invisible in assembly drawings	1. In the **Assembly Drawing Properties** dialog box, click **Bolts**. 2. On the **Content** tab, select **solid** from the **Solid/Symbol** list. 3. Click **OK**. 4. Go to **File menu** --> **Settings** --> **Advanced options** --> **Drawing Properties** , and set `XS_DRAW_BOLT_HIDDEN_LINES_IN_ASSEMBLY_DRAWINGS` to `FALSE`. 5. Create the drawing.
Set the hidden lines to be visible as the assembly drawing part properties allow	1. In the **Assembly Drawing Properties** dialog box, click **Bolts**. 2. On the **Content** tab, select **solid** from the **Solid/Symbol** list. 3. Click **OK**. 4. Click **Part** in the drawing properties dialog box.

To	Do this
	5. On the **Content** tab, remove the check mark from the **Hidden line on/off** check box. 6. Go to **File menu** --> **Settings** --> **Advanced options** --> **Drawing Properties** , and set XS_DRAW_BOLT_HIDDEN_LINES_IN_ASSEMBLY_DRAWINGS to AS_PART. 7. Create the drawing.

See also

XS_DRAW_BOLT_OWN_HIDDEN_LINES (page 185)

XS_DRAW_BOLT_HIDDEN_LINES_IN_ASSEMBLY_DRAWINGS

Category

Drawing Properties

Use this advanced option to show or hide bolts that are hidden from view by other parts in assembly drawings.

The possible values are:

- AS_PART: uses part's hidden line visibility settings.
- TRUE: always shows hidden bolts.
- FALSE: does not show hidden bolts.

The default value is AS_PART.

NOTE This setting only affects bolts with solid or exact solid representation. Bolts with symbol representation are always visible. Changing the advanced option does not affect existing drawings, but you need to recreate them.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

See also

XS_DRAW_BOLT_HIDDEN_LINES (page 181).

XS_DRAW_BOLT_HIDDEN_LINES_IN_GA_DRAWINGS

Category

Drawing Properties

Use this advanced option to show or hide bolts that are hidden from view by other parts in general arrangement drawings.

The possible values are:

- `AS_PART`: uses part's hidden line visibility settings.
- `TRUE`: always shows hidden bolts.
- `FALSE`: does not show hidden bolts.

The default value is `FALSE`.

NOTE This setting only affects bolts with solid or exact solid representation. Bolts with symbol representation are always visible. Changing the advanced option does not affect existing drawings, but you need to recreate them.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

See also

XS_DRAW_BOLT_HIDDEN_LINES (page 181).

XS_DRAW_BOLT_HIDDEN_LINES_IN_SINGLE_DRAWINGS

Category

Drawing Properties

Use this advanced option to show or hide bolts that are hidden from view by other parts in single part drawings.

The possible values are:

- `AS_PART`: uses part's hidden line visibility settings.
- `TRUE`: always shows hidden bolts.
- `FALSE`: does not show hidden bolts.

The default value is `AS_PART`.

NOTE This setting only affects bolts with solid or exact solid representation. Bolts with symbol representation are always visible. Changing the

advanced option does not affect existing drawings, but you need to recreate them.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DRAW_BOLT_HIDDEN_LINES (page 181).

XS_DRAW_BOLT_OWN_HIDDEN_LINES

Category

Drawing properties

Use the following advanced options show or hide own hidden lines in bolts in single part, assembly, and general arrangement drawings. The *own hidden lines* are the lines of the object representation covered up by the object itself.

The default values are shown here:

- `XS_DRAW_BOLT_OWN_HIDDEN_LINES_IN_SINGLE_DRAWINGS=AS_PART`
- `XS_DRAW_BOLT_OWN_HIDDEN_LINES_IN_ASSEMBLY_DRAWINGS=AS_PART`
- `XS_DRAW_BOLT_OWN_HIDDEN_LINES_IN_GA_DRAWINGS=FALSE`

Possible values are:

- `AS_PART`: used the visibility settings of the part's own hidden lines.
- `TRUE`: always shows bolts' hidden lines.
- `FALSE`: does not show bolts' hidden lines.

See also

XS_DRAW_BOLT_HIDDEN_LINES (page 181)

XS_DRAW_BOLT_OWN_HIDDEN_LINES_IN_ASSEMBLY_DRAWINGS

Category

Drawing Properties

To show or hide the hidden lines of the bolts based on the own hidden line settings of the part that hides the bolts, enter `AS_PART`. To always show

hidden lines of the hidden bolts, enter TRUE. To never show the hidden lines, enter FALSE. The default value is AS_PART.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

See also

XS_DRAW_BOLT_OWN_HIDDEN_LINES (page 185)

XS_DRAW_BOLT_OWN_HIDDEN_LINES_IN_GA_DRAWINGS

Category

Drawing Properties

To show or hide the hidden lines of the bolts based on the own hidden line settings of the part that hides the bolts, enter AS_PART. To always show hidden lines of the hidden bolts, enter TRUE. To never show hidden lines of the hidden bolts, enter FALSE. The default value is FALSE.

NOTE This setting only affects bolts with solid or exact solid representation. Bolts with symbol representation are always visible. Changing the advanced option does not affect existing drawings, but you need to recreate them.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

See also

XS_DRAW_BOLT_OWN_HIDDEN_LINES (page 185)

XS_DRAW_BOLT_OWN_HIDDEN_LINES_IN_SINGLE_DRAWINGS

Category

Drawing Properties

To show or hide the hidden lines of the bolts based on the own hidden line settings of the part that hides the bolts, enter AS_PART. To always show hidden lines of the hidden bolts, enter TRUE. To never show hidden lines of the hidden bolts, enter FALSE. The default value is AS_PART.

NOTE This setting only affects bolts with solid or exact solid representation. Bolts with symbol representation are always visible. Changing the advanced option does not affect existing drawings, but you need to recreate them.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DRAW_BOLT_OWN_HIDDEN_LINES (page 185)

XS_DRAW_BOLTS_3D_IN_BOLT_LAYER

Category: Model View

When this advanced option is set to `TRUE`, which is the default value, the model views that display bolts as rendered solid objects (using the **Exact** representation option) open and update faster. This is especially useful in large models with steel structures.

This advanced option is system specific and is read from environment files. It can also be read from `teklastructures.ini`. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

After changing the value of this advanced option, first save the model. Then restart Tekla Structures to activate the new setting.

XS_DRAW_BOLTS_PERPENDICULAR_TO_PART_IN_SINGLE_DRAWINGS

Category

Dimensioning: Bolts

Set this advanced option to `TRUE` to draw bolts perpendicular to the part plane in single part drawings. If you do not want to do this, set it to `FALSE`. This only applies to bolt symbol types **symbol** and **symbol3**.

This advanced option is model specific and the setting is saved in the options database.

XS_DRAW_BOLTS_THROUGH_NEIGHBOUR_PARTS

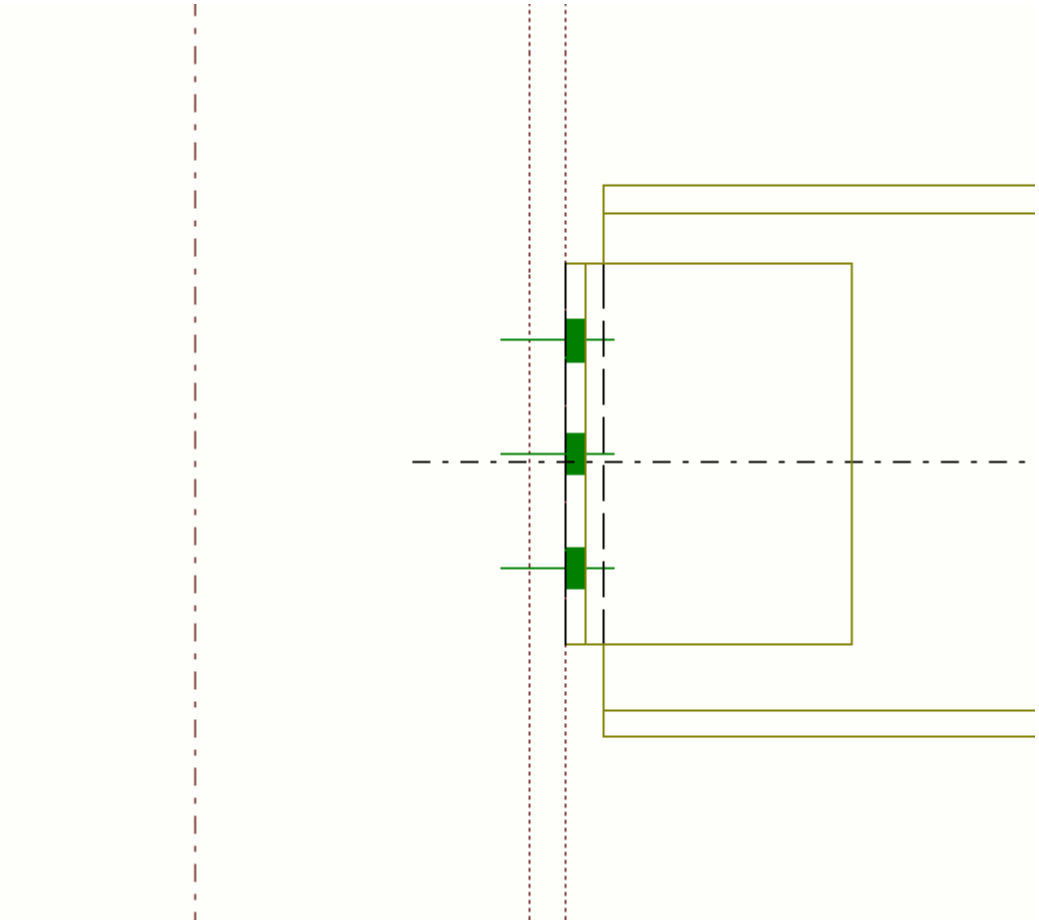
Category

Drawing Properties

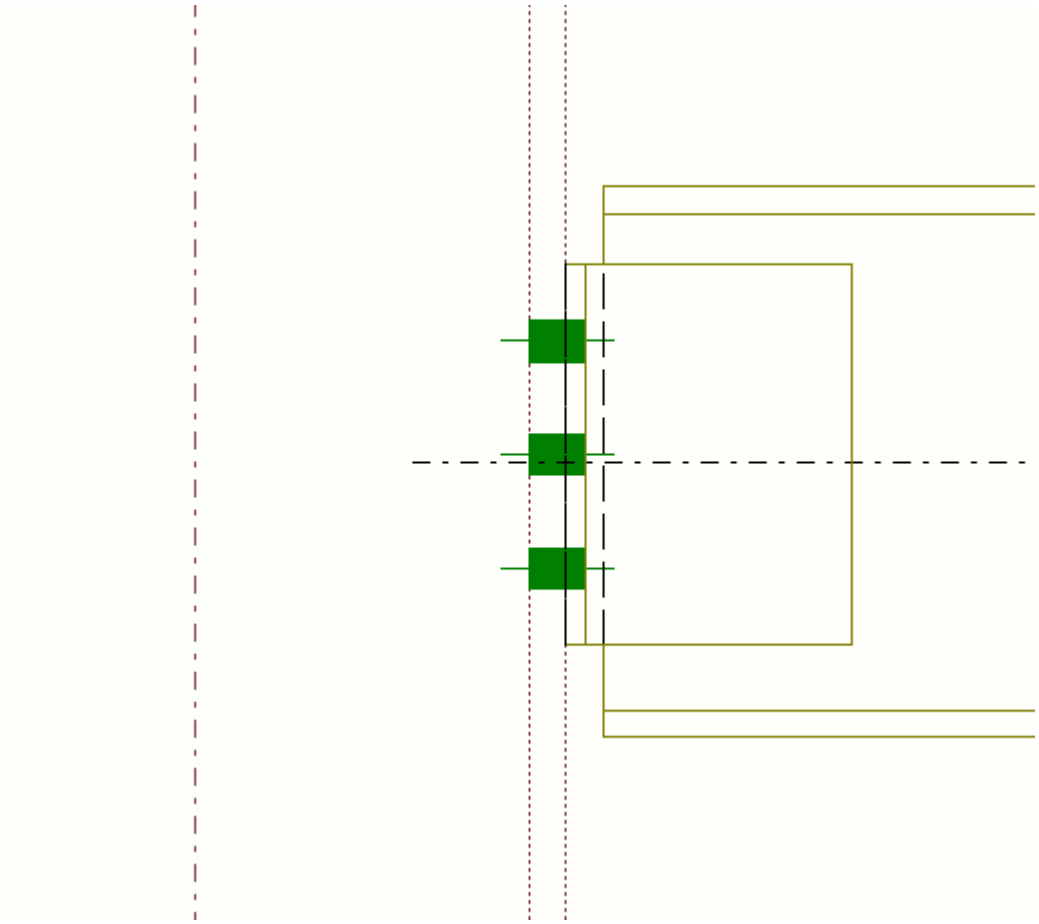
Set this advanced option to `TRUE` to draw the bolt parts that coincide with neighboring parts. If you do not want to draw these bolt parts, set it to `FALSE`. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

The following example shows how the drawing looks like when you set this advanced option to `FALSE`.



In the following example, the advanced option is set to `TRUE`.



XS_DRAW_CAST_PHASE_INTERNAL_LINES

Category: Concrete detailing

Use to show or hide edge lines of (precast) concrete cast units in drawings. Set to `TRUE` (default) to show overlapping edge lines between adjacent cast units that are in the same phase.

NOTE If you are working with cast-in-place concrete parts and `XS_ENABLE_POUR_MANAGEMENT` is set to `TRUE`, the advanced options `XS_DRAW_CAST_UNIT_INTERNAL_LINES` and `XS_DRAW_CAST_PHASE_INTERNAL_LINES` do not have an effect.

If `XS_ENABLE_POUR_MANAGEMENT` is set to `FALSE`, both cast-in-place and precast concrete parts are treated in the same way and are affected by the advanced options

XS_DRAW_CAST_UNIT_INTERNAL_LINES and XS_DRAW_CAST_PHASE_INTERNAL_LINES.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

See also

XS_DRAW_CAST_UNIT_INTERNAL_LINES (page 190)

XS_DRAW_CAST_UNIT_INTERNAL_LINES

Category: Concrete Detailing

Use to show or hide concrete part lines inside (precast) cast units in drawings. Set to TRUE to show overlapping part lines inside cast units.

The default value is FALSE.

NOTE If you are working with cast-in-place concrete parts and XS_ENABLE_POUR_MANAGEMENT is set to TRUE, the advanced options XS_DRAW_CAST_UNIT_INTERNAL_LINES and XS_DRAW_CAST_PHASE_INTERNAL_LINES do not have an effect.

If XS_ENABLE_POUR_MANAGEMENT is set to FALSE, both cast-in-place and precast concrete parts are treated in the same way and are affected by the advanced options XS_DRAW_CAST_UNIT_INTERNAL_LINES and XS_DRAW_CAST_PHASE_INTERNAL_LINES.

This advanced option is model specific and the setting is saved in the options database.

See also

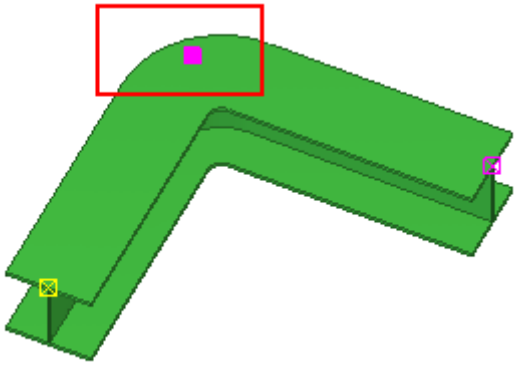
XS_DRAW_CAST_PHASE_INTERNAL_LINES (page 189)

XS_DRAW_CHAMFERS_HANDLES

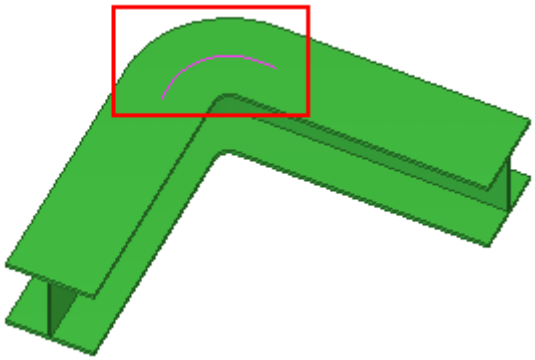
Category: Model view

Use to define whether to show the handles or the chamfers for contour plates, concrete slabs, and polybeams. The options are:

- HANDLES: handles are shown. Makes the selection of handles easier. This is the default value.



- CHAMFERS: chamfers are shown. Use this option, for example, when you want to check the status of polybeam chamfers.



- CHAMFERS_AND_HANDLES shows both the chamfers and handles.

See also

XS_DO_NOT_DISPLAY_CHAMFERS (page 173)

XS_DRAW_CROSS_AXIS

Category

Drawing properties

Set to N to have Tekla Structures hide the axis cross in beam cross sections.

Leave the value out to display the axis cross in beam cross sections.

This advanced option is model specific and the setting is saved in the options database.

XS_DRAW_CUT_FACES_WITH_OBJECT_COLOR

Category: Model View

Set to `FALSE` (default) to have cut faces shown in gray, and to `TRUE` to show the cut faces with the same color as the other object faces.

Redraw the model view after changing the value for the change to take effect.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`.

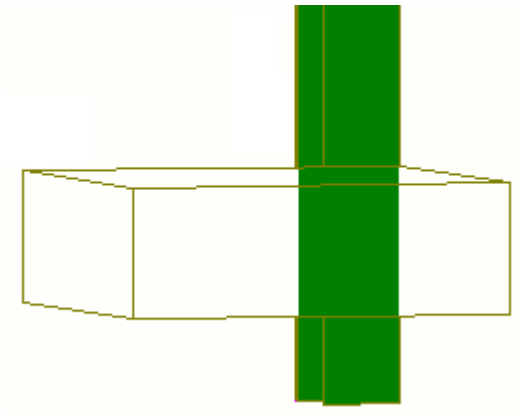
XS_DRAW_HIDDEN_FACES

Category: Drawing Properties

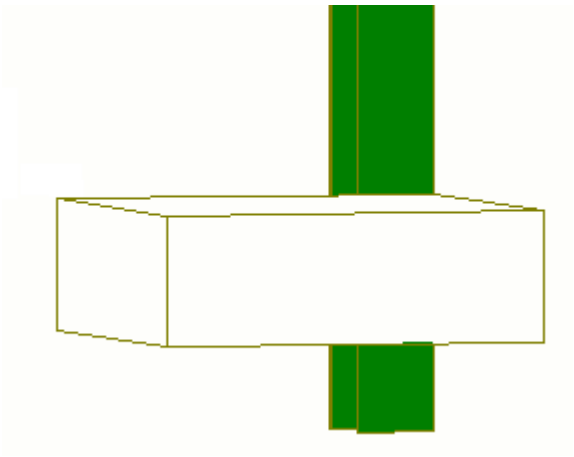
Set this advanced option to `TRUE` to draw hidden part faces in drawings. The default value is `FALSE`. This advanced option hides the hatches on the hidden part faces, for example.

This advanced option is model specific and the setting is saved in the options database.

Example of using the value `TRUE`:



Example of using the value `FALSE`:



TIP To hide hidden faces of rebars, use the advanced option XS_DRAW_REBAR_HIDDEN_FACES (page 194).

XS_DRAW_HORIZONTAL_VIEW_SHORTENING_SYMBOLS_TO_PARTS

Category

Drawing Properties

Set to `TRUE` to show horizontal view shortening symbols automatically. The default value is `FALSE`.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

Limitations

- Shortening symbols are not shown in general arrangement drawings.
- Shortening symbols are not shown in rectangular hollow sections or channel sections.

See also

XS_DRAW_VERTICAL_VIEW_SHORTENING_SYMBOLS_TO_PARTS (page 199)

XS_SHORTENING_SYMBOL_WITH_ZIGZAG (page 416)

XS_SHORTENING_SYMBOL_COLOR (page 415)

XS_SHORTENING_SYMBOL_LINE_TYPE (page 415)

XS_DRAW_INSIDE_ANGLE_IN_UNFOLDING

Category

Dimensioning: Unfolding

Set this advanced option to `TRUE` to display the interior angle instead of the exterior angle in angle text. If set this advanced option to `FALSE` (default), the exterior angle is displayed.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DRAW_LONG_HOLE_DIMENSIONS

Category

Dimensioning: Bolts

Set this advanced option to `TRUE` to dimension slotted holes to the center points of the curve. If you set this advanced option to `FALSE` (default), the slotted holes are dimensioned to the center points of the holes.

This advanced option is model specific and the setting is saved in the options database.

XS_DRAW_MESH_OUTLINE_SYMBOL _FROM_BOTTOM_LEFT_TO_TOP_RIGHT

Category

Concrete Detailing

Use to define the outline representation of reinforcement meshes in drawings. When you set this advanced option to `TRUE` (default), the outline representation symbol of the mesh is always from bottom left to top right.

This advanced option is model specific and the setting is saved in the options database.

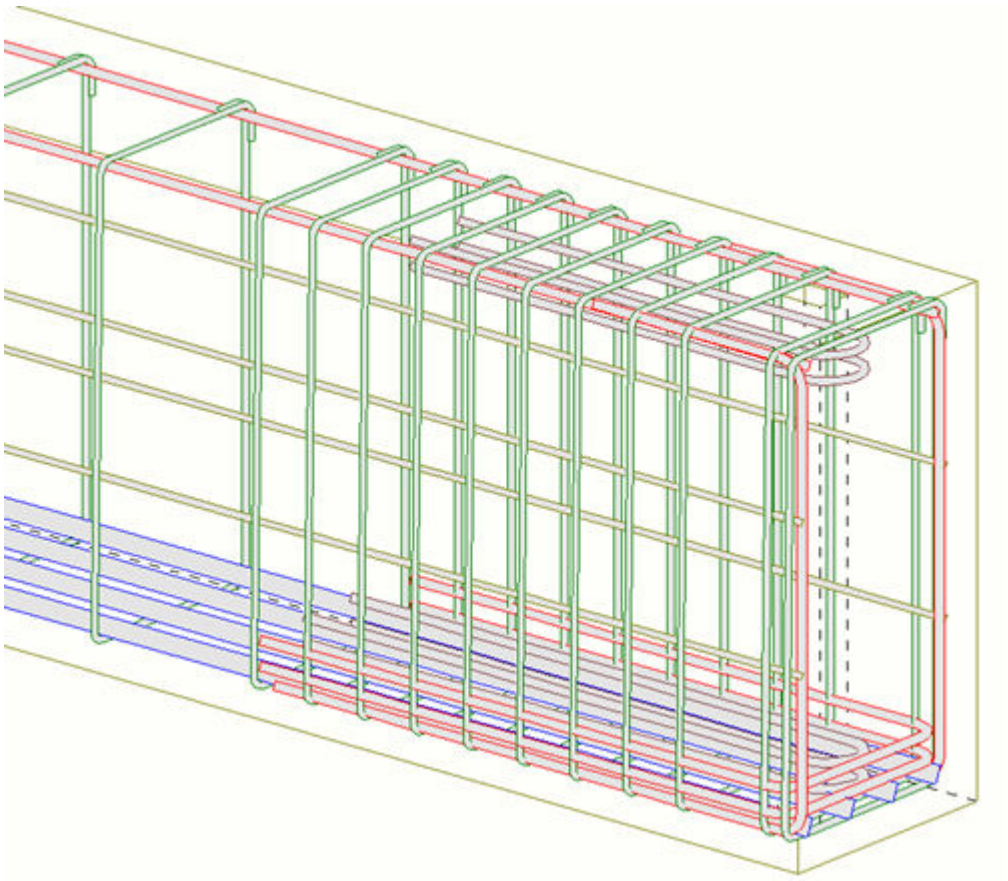
XS_DRAW_REBAR_HIDDEN_FACES

Category: **Drawing Properties**

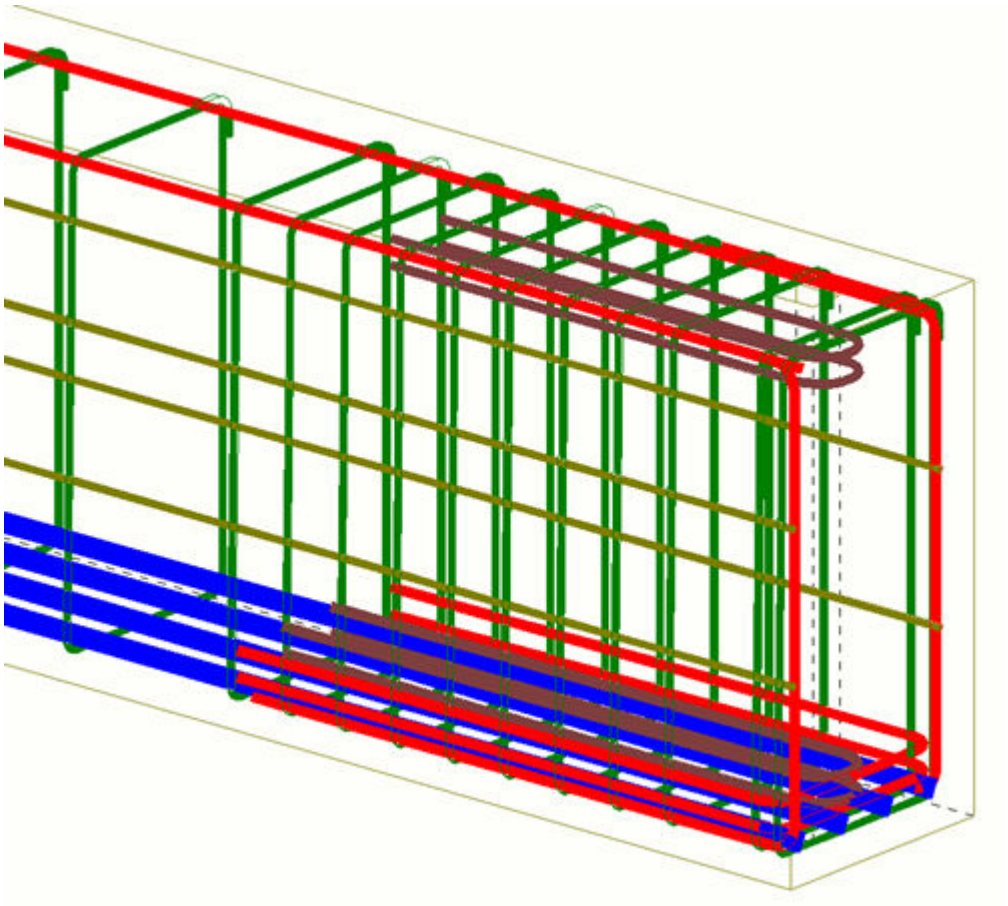
Use this advanced option to control the display of the rebar hidden faces in drawings. To show the hidden faces of the rebars, set this advanced option to `TRUE`. `FALSE` is the default value.

This advanced option is model specific and the setting is saved in the options database.

In the first image below, `XS_DRAW_REBAR_HIDDEN_FACES` has been set to `TRUE`.



In the image below, `XS_DRAW_REBAR_HIDDEN_FACES` has been set to `FALSE`.



This advanced option is model specific and the setting is saved in the options database.

See also

XS_DRAW_HIDDEN_FACES (page 192)

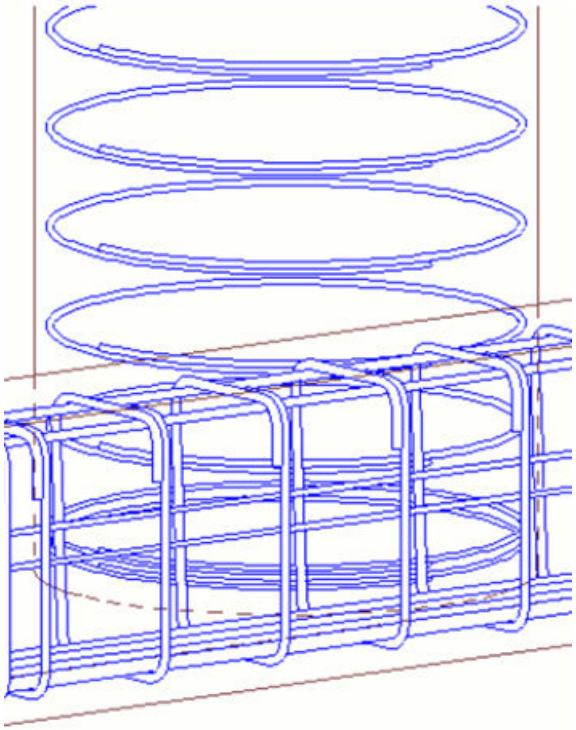
XS_DRAW_REBAR_SELF_INTERSECTING_LEGS_WITH_OFFSET

Category: Drawing Properties

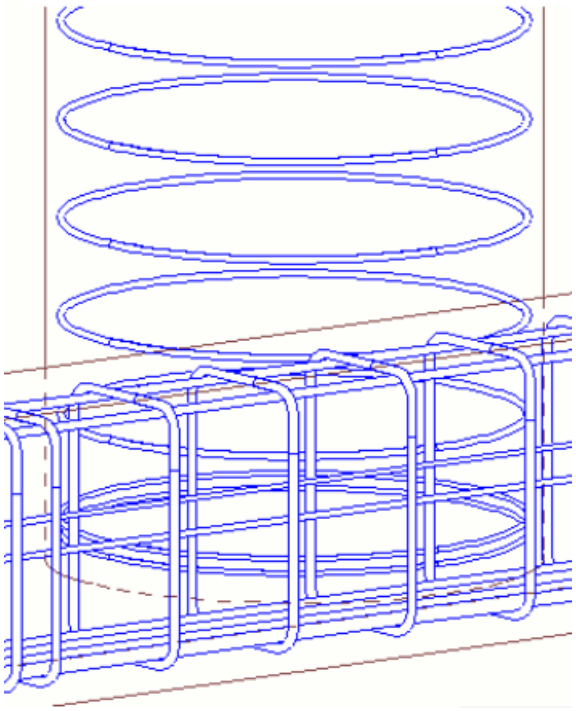
When set to `TRUE`, this advanced option displays rebars in drawings with overlap in all representations. Note that `FALSE` has no impact on the **filled line** representation. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

`XS_DRAW_REBAR_SELF_INTERSECTING_LEGS_WITH_OFFSET` set to `TRUE`:



XS_DRAW_REBAR_SELF_INTERSECTING_LEGS_WITH_OFFSET set to FALSE:



See also

Reinforcement object properties in drawings (page 768)

XS_DRAW_ROOT_OPENING_EVEN_WHEN_ZERO

Category

Welds

Set this advanced option to `TRUE` to show zero root openings. The default is `TRUE`. If you set this advanced option to `FALSE`, the zero root openings are not shown.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_DRAW_SHORT_LEADER_LINES_OF_PART_MARKS

Category

Marking: Parts

Use this advanced option to define if a leader line is drawn when the leader line is shorter than defined with the advanced option `XS_DRAW_SHORT_LEADER_LINES_OF_PART_MARKS_MINIMUM_LENGTH`. When you set this option to `FALSE`, the leader line is not drawn. Set to `TRUE` (default) to always draw leader lines in part marks.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DRAW_SHORT_LEADER_LINES_OF_PART_MARKS_MINIMUM_LENGTH (page 198)

XS_DRAW_SHORT_LEADER_LINES _OF_PART_MARKS_MINIMUM_LENGTH

Category

Marking: Parts

Use this advanced option to define the minimum length of a leader line that Tekla Structures draws. If the leader line is shorter that the minimum length and the advanced option `XS_DRAW_SHORT_LEADER_LINES_OF_PART_MARKS` is set to `FALSE`, the leader line is not drawn. The default value is `0.0`. If the advanced option

XS_DRAW_SHORT_LEADER_LINES_OF_PART_MARKS_MINIMUM_LENGTH is set to TRUE, the part mark leader lines are always drawn.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DRAW_SHORT_LEADER_LINES_OF_PART_MARKS (page 198)

XS_DRAW_SKEWED_ELEVATIONS

Category

Dimensioning: Parts

Set this advanced option to TRUE to display skewed elevation dimensions.

If you set it to FALSE (default), skewed elevation dimensions are not displayed in drawings.

This advanced option is model specific and the setting is saved in the options database.

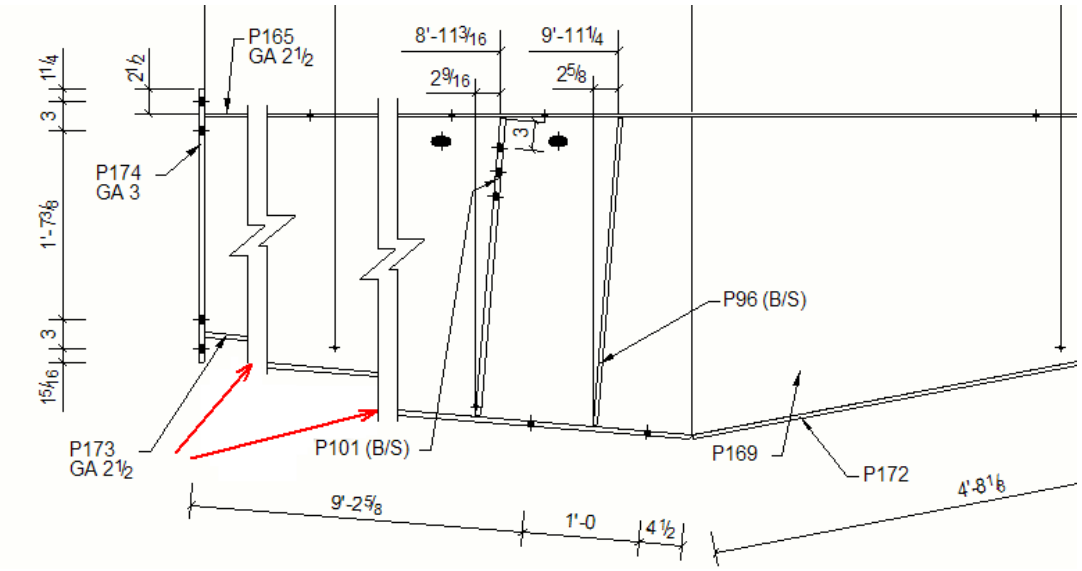
See also

XS_DRAW_VERTICAL_VIEW_SHORTENING_SYMBOLS_TO_PARTS

Category

Drawing Properties

Set this advanced option to TRUE to show vertical view shortening symbols automatically. The default value is FALSE.



This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

Limitations

- Shortening symbols are not shown in general arrangement drawings.
- Shortening symbols are not shown in rectangular hollow sections or channel sections.

See also

XS_DRAW_HORIZONTAL_VIEW_SHORTENING_SYMBOLS_TO_PARTS (page 193)

XS_SHORTENING_SYMBOL_WITH_ZIGZAG (page 416)

XS_SHORTENING_SYMBOL_COLOR (page 415)

XS_SHORTENING_SYMBOL_LINE_TYPE (page 415)

XS_DRAWING_ALLOW_NEW_SECTIONS_IN_REDIMENSIONING

Category

Drawing Properties

Use to define whether new views or sections are created during redimensioning of existing drawings. The default is `FALSE`, which means that no views or sections are created.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is

in use, you can change the value, which is then the same for all users in the current model.

XS_DRAWING_ALLOW_SNAPPING_TO_DISTANT_POINTS

Category

Drawing Properties

Set this advanced option to `TRUE` to allow the cursor to snap to object end points in a drawing even if the cursor is not near the end point. This means that if the cursor is anywhere on the object, the cursor will snap to the object end points. If you do not want to allow this, set this advanced option to `FALSE`. Then the cursor will snap only to the snap points near the cursor.

The default value is `FALSE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_DRAWING_ASSEMBLY_HATCH_SCHEMA

Category

Hatching

Use to indicate the name of the schema file to use for assembly drawings.

Example

To use the default schema file, enter `assembly.htc`.

This advanced option is model specific and the setting is saved in the options database.

XS_DRAWING_CAST_UNIT_HATCH_SCHEMA

Category

Hatching

Use to determine the name of the schema file used for cast unit drawings.

This advanced option is model specific and the setting is saved in the options database.

Example

To use the default schema file, enter `cast_unit.htc`.

XS_DRAWING_CHANGE_HIGHLIGHT_COLOR

Category

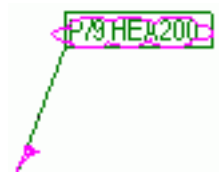
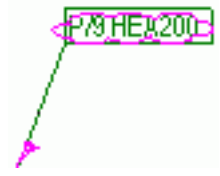
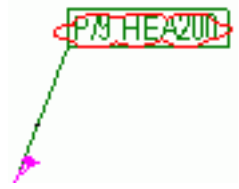
Drawing Properties

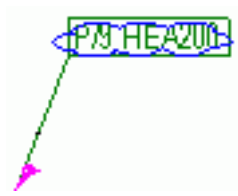
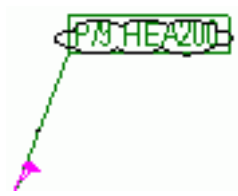
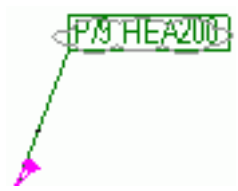
Use to change the highlight color of the automatic change symbols. The color options are `RED`, `DARK RED`, `ORANGE`, `DARK YELLOW`, `GREEN`, `DARK GREEN`, `BLUE`, `DARK BLUE`, `BLACK`, `GREY`, `DARK GREY`, `CYAN`, `DARK CYAN` and `MAGENTA`. You can also enter colors as numerical values.

NOTE If you use the default color 190, the symbols are displayed on the screen but they will not appear in the printed drawings.

This advanced option is model specific and the setting is saved in the options database.

Example

Set the advanced option to	Color on screen	Color in printed drawings
`190 (default)`		not visible
`MAGENTA`		magenta
`RED`		red
`GREEN`		green

Set the advanced option to	Color on screen	Color in printed drawings
BLUE		blue
BLACK		black
GREY		grey

XS_DRAWING_CLONING_IGNORE_CHECK

Category: **Drawing Properties**

Set to TRUE to have Tekla Structures clone a drawing even though all the parts of the original drawing have been deleted and the position number is the same as in the original drawing. The default is FALSE.

Note that when XS_DRAWING_CLONING_IGNORE_CHECK is set to TRUE, Tekla Structures is able to clone a drawing with a selected part that has the same position number. But if you have selected several parts with the same position number, then only one cloned drawing is created for that position.

This advanced option is model specific and the setting is saved in the options database.

Example

You have drawing A[A.1] in your **Document manager**. You number the model so that the assembly A.1 changes to A.2. In **Document manager**, the A[A.1] drawing is marked with an x and with the status message "All parts deleted." Then you number the model again, so that the assembly in the model changes from A.2 back to A.1. To clone, set the advanced option XS_DRAWING_CLONING_IGNORE_CHECK to TRUE, select the A[A.1] drawing (marked with x) from **Document manager** and assembly A.1 from the model, and click **Clone**.

XS_DRAWING_COMBINE_ADDED_DIMENSIONS

Category

Drawing Properties

Use to control whether added dimensions are merged to existing ones in updated or cloned drawings. If set to `FALSE`, added dimensions are not merged to existing ones. The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

XS_DRAWING_CUT_VIEW_COMPARISON_CRITERIA

Category: Drawing View

Use to specify the criteria used when comparing section views. If section views are different, they will be visible and will receive a unique section mark. By default, section views are compared based on the boundaries of the parts (`EXTREMA`) and the orientation of the parts in the view (`ORIENTATION`). The advanced option may contain a combination of the following options separated by a comma:

- `POSITION` compares position numbers of all the parts in the view (including non-dimensioned parts).
- `EXTREMA` compares the boundaries of the parts in the view.
- `ORIENTATION` compares the orientation of the parts in the view.
- `SHOWALL` considers all section views different and displays them with unique section marks.
- `EXACT` uses more rigid rules when comparing section views. Use `EXACT` only in combination with the options `EXTREMA` or `ORIENTATION`.

  `EXACT`, `ORIENTATION` = Part x/y axes need to be the same.

  `ORIENTATION` = Part x/y axes need to be the same or opposite.

  `EXACT`, `EXTREMA` = Part extremas need to be the same in any direction.

  `EXTREMA` = Part extremas need to be the same in each part's local direction.

  If orientation is 90 degrees different, the extremas can still be the same if both parts have the same x and y dimension, for example.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_DRAWING_FILTER_UDAS_WITHOUT_TYPE_CHECK

Category

Drawing Properties

Set this advanced option to `FALSE` (default) to use only user-defined attributes that have been defined for the object in the `object.inp` in drawing view filtering.

This advanced option is model specific and the setting is saved in the options database.

XS_DRAWING_GA_HATCH_SCHEMA

Category

Hatching

Use to determine the name of the schema file used for general arrangement drawings.

This advanced option is model specific and the setting is saved in the options database.

Example

To use the default schema file, enter `general.htc`.

XS_DRAWING_GRID_LABEL_FRAME_FIXED_WIDTH

Category

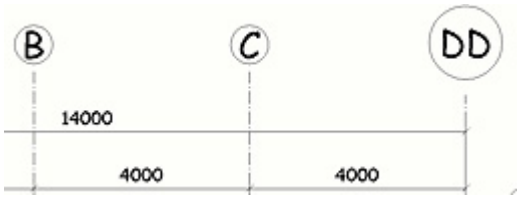
Drawing Properties

Use to specify a fixed size for the grid label frames. This is useful, for example, when you want all the grid label frames to be the same size no matter whether there are one or two digits in the frame. If you set this advanced option to zero (0), the width of the grid label frame depends on the width of the grid label. Enter the desired value in millimeters.

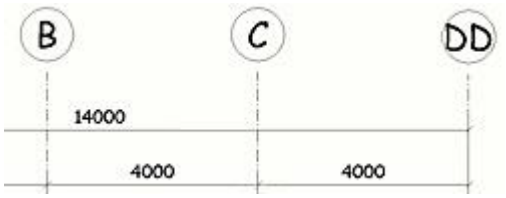
A fixed width of 18 is suggested for 5 characters (XX.XX) when the text height is 3/16. Change the fixed width to 14 for 4 characters (XX.X), 12 for 3 characters (X.X), and to 10 for 2 characters (XX). If you use a different text height than 3/16, fixed width values need to be adjusted accordingly. This advanced option overrides the automatic frame width calculation for grid labels.

If you do not set this advanced option, Tekla Structures adjusts the grid label frames to text within each frame.

Example of grid label frames when the frame size is not fixed:



Example of grid label frames when the frame size is fixed:



This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_DRAWING_GRID_LABEL_FRAME_LINE_WIDTH_FACTOR

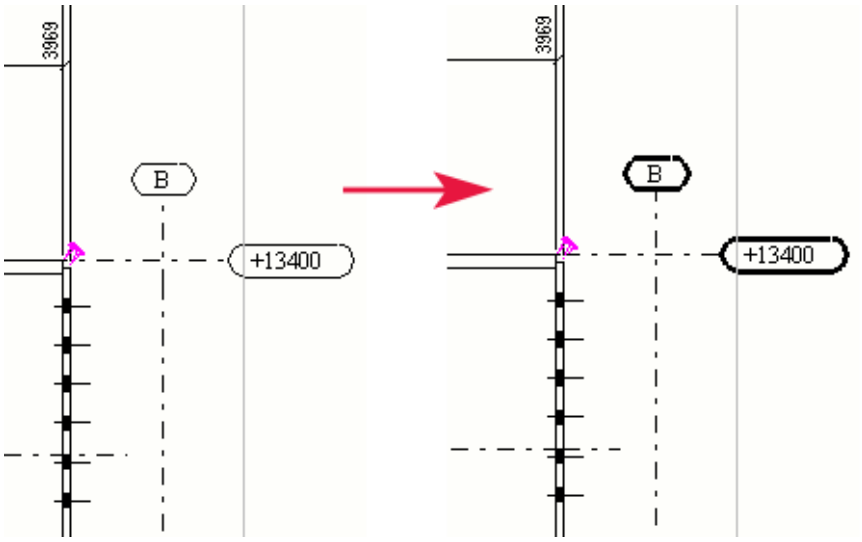
Category

Drawing Properties

Use to change the thickness of the grid label frame in drawings. You may want to emphasize the grid label frames so that they are shown with a thicker line than the rest of the grid.

Example

```
XS_DRAWING_GRID_LABEL_FRAME_LINE_WIDTH_FACTOR=1
```



NOTE Each color has a certain line thickness. The thickness of the grid label frame on the printed black-and-white drawings depends on the color that has been defined for the grid label in the grid properties and on the value of this advanced option. .

This advanced option is model specific and the setting is saved in the options database.

XS_DRAWING_HISTORY_LOG_TYPE

Category

Drawing Properties

Use to define the contents of the drawing history log file `drawing_history.log`. You can use the following options individually or in any combination.

- `ALL`
- `NEW`
- `DELETED` (default)
- `MODIFIED`

Example

Separate the options using the _ character, for example, `NEW_DELETED`.

This advanced option is model specific and the setting is saved in the options database.

XS_DRAWING_IGNORE_ZERO_LEVELS_IN_PART_MARKS

Category

Marking: Parts

Use to control whether zero levels (+0.000) are shown or hidden in part marks. By default the advanced option is set to `FALSE`, so zero levels are shown in part marks. Set to `TRUE` to hide the zero levels in part marks.

For example, you can use this advanced option for hiding connection side marks at the specified distance from the view plane.

TIP To list level information in part marks, go to the **Part mark properties** dialog box and insert the element **User-defined attribute** and enter one of the following template attributes:

- `ASSEMBLY_BOTTOM_LEVEL`
- `ASSEMBLY_TOP_LEVEL`
- `CAST_UNIT_BOTTOM_LEVEL`
- `CAST_UNIT_TOP_LEVEL`

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

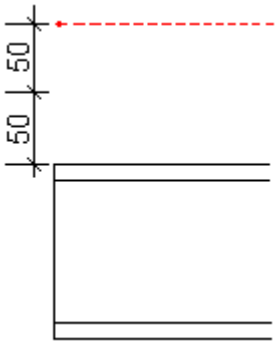
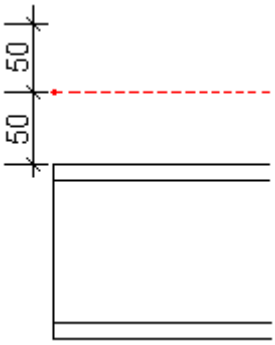
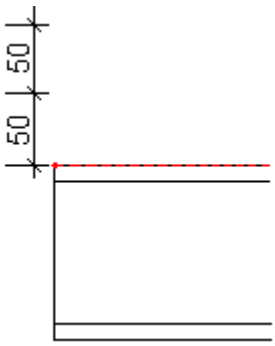
See also

XS_DRAWING_PART_REFERENCE_LINE_TYPE

Category:Drawing Properties

Use to set the reference line type in drawings. You can use the following switches:

- `POINT_LINE` creates a line between the part's creation points.
- `DEFINITION_LINE` creates a line between the part's definition points (creation points + end offsets).
- `CORNER_REFERENCE_LINE` creates a line between the part's corner points.

POINT_LINE	DEFINITION_LINE	CORNER_ REFERENCE_LINE
		

This advanced option is model specific and the setting is saved in the options database.

XS_DRAWING_PART_SYMBOL_REPRESENTATION_TYPE

Category

Drawing Properties

Use this advanced option for adjusting the symbol representation of parts in drawings. Value `0` (default) means by reference line, and value `1` means by center line. This affects the part representation options **Symbol** and **Symbol with partial profile** in the part properties dialog box.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DRAWING_PLOT_FILE_DIRECTORY

Category Printing

Use this advanced option to define the folder where **Printer Catalog** creates the plot files if the file name field is empty in the **Print Drawings** dialog box. By default, `.\PlotFiles` is used.

Note that **Printer Catalog** is used only when you have set the advanced option `XS_USE_OLD_PLOT_DIALOG` to TRUE in **File menu** --> **Settings** --> **Advanced options** --> **Printing** .

This advanced option also defines the folder where new DWG/DXF drawing export creates dwg/dxf files if the file location field is empty in the **Export drawings to DWG/DXF** dialog box. By default, .\ is used.

NOTE This advanced option overrides the folder defined in the **Printer Catalog** and in the new **Export drawings to DWG/DXF** dialog box.

XS_DRAWING_PLOT_FILE_NAME

This advanced option must be set in an initialization (.ini) file.

Use this advanced option to define print file names for drawings, if the filename is missing from the **Print Drawings** dialog box. This advanced option is used if you have not entered a value for some of the following advanced options: XS_DRAWING_PLOT_FILE_NAME_A, XS_DRAWING_PLOT_FILE_NAME_C, XS_DRAWING_PLOT_FILE_NAME_G, XS_DRAWING_PLOT_FILE_NAME_W, or XS_DRAWING_PLOT_FILE_NAME_M.

Enter any combination of text and options:

NAME

NAME.-

NAME.

DRAWING_NAME

DRAWING_NAME.

DRAWING_NAME.-

REVISION

DRAWING_REVISION

REV_MARK

REVISION_MARK

DRAWING_REVISION_MARK

REV

TITLE

DRAWING_TITLE

UDA:<drawing user-defined attribute>

TPL:<template attribute>

<variable>?- <text>

The question mark (?) can be used for querying if a <variable> is set, and if it does, the <text> after the question mark until the last % will be printed. See the second example below.

This advanced option is model specific and the setting is saved in the options database.

Examples

```
%DRAWING_NAME.% - %DRAWING_TITLE%%DRAWING_REVISION? - Rev %
%REVISION_MARK%
```

```
%NAME% - %TITLE%%UDA:DRAWING_USERFIELD_1? - %
%UDA:DRAWING_USERFIELD_1%%DRAWING_REVISION? - Rev%
%DRAWING_REVISION%
```

If you define `%DRAWING_NAME.% - %DRAWING_TITLE%
%DRAWING_REVISION? - Rev %%REVISION_MARK%`, a .pdf file with a name `P1 - PLATE - Rev A.pdf` will be created from a single part drawing, for example, where

`DRAWING_NAME.` = P1, this is the part position without punctuation.

`DRAWING_TITLE` = PLATE, this is the name that you have entered in the **Name** box in drawing properties.

`DRAWING_REVISION` = empty, if there are no revisions, or a numeric value 1, 2, 3, etc according to the revision selected in **Rev.No.** in the **Revision Handling** dialog box. The actual value is not written in the plot file name, but the text after the question mark (?).

`- Rev` is the text to be printed, if `DRAWING_REVISION` gives a value. If `DRAWING_REVISION` does not give a value, the text `- Rev` will not be printed. In this case, the plot file name will not have `REVISION_MARK` either because the drawing has not been revised.

`REVISION_MARK` = A, because A has been defined as the revision mark in the **Revision Handling** dialog box.

See also

Customize print file names

XS_DRAWING_PLOT_FILE_NAME_A

Category: **Printing**

Use to define print file names for assembly drawings.

Enter any combination of text and options:

`NAME`

`NAME.-`

NAME.

DRAWING_NAME

DRAWING_NAME.

DRAWING_NAME.-

REVISION

DRAWING_REVISION

REV_MARK

REVISION_MARK

DRAWING_REVISION_MARK

REV

TITLE

DRAWING_TITLE

UDA:<drawing user-defined attribute>

TPL:<template attribute>

<variable>?- <text>

The question mark (?) can be used for querying if a <variable> is set, and if it does, the <text> after the question mark will be printed. See the second example below.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

Example

```
%DRAWING_NAME.% - %DRAWING_TITLE%%DRAWING_REVISION? - Rev %
%REVISION_MARK%
```

```
%NAME% - %TITLE%%UDA:DRAWING_USERFIELD_1? - %
%UDA:DRAWING_USERFIELD_1%%DRAWING_REVISION? - Rev%
%DRAWING_REVISION%
```

See also

Customize print file names

XS_DRAWING_PLOT_FILE_NAME_W

Category: **Printing**

Use to define print filenames for single part drawings. Enter any combination of text and options: NAME, NAME.-, NAME., DRAWING_NAME, DRAWING_NAME.,

DRAWING_NAME.-, REVISION, DRAWING_REVISION, REV_MARK, REVISION_MARK, DRAWING_REVISION_MARK, REV, TITLE, DRAWING_TITLE, UDA:<drawing user-defined attribute>, TPL:<template attribute>, <variable>?- <text>

The question mark (?) can be used for querying if a <variable> is set, and if it does, the <text> after the question mark will be printed. See the second example below.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

Examples

```
%DRAWING_NAME.% - %DRAWING_TITLE%%DRAWING_REVISION? - Rev %
%REVISION_MARK%
```

```
%NAME% - %TITLE%%UDA:DRAWING_USERFIELD_1? - %
%UDA:DRAWING_USERFIELD_1%%DRAWING_REVISION? - Rev%
%DRAWING_REVISION%
```

See also

Customize print file names

XS_DRAWING_PLOT_FILE_NAME_G

Category: **Printing**

Use to define print file names for general arrangement drawings.

Enter any combination of text and options:

NAME

NAME.-

NAME.

DRAWING_NAME

DRAWING_NAME.

DRAWING_NAME.-

REVISION

DRAWING_REVISION

REV_MARK

REVISION_MARK

DRAWING_REVISION_MARK

REV

TITLE

DRAWING_TITLE

UDA:<drawing user-defined attribute>

TPL:<template attribute>

<variable>?- <text>

The question mark (?) can be used for querying if a <variable> is set, and if it does, the <text> after the question mark will be printed. See the second example below.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

Example

```
%DRAWING_NAME.% - %DRAWING_TITLE%%DRAWING_REVISION? - Rev %
%REVISION_MARK%
```

```
%NAME% - %TITLE%%UDA:DRAWING_USERFIELD_1? - %
%UDA:DRAWING_USERFIELD_1%%DRAWING_REVISION? - Rev%
%DRAWING_REVISION%
```

See also

Customize print file names

XS_DRAWING_PLOT_FILE_NAME_M

Category: **Printing**

Use to define print file names for multidrawings.

Enter any combination of text and options:

NAME

NAME.-

NAME.

DRAWING_NAME

DRAWING_NAME.

DRAWING_NAME.-

REVISION

DRAWING_REVISION

REV_MARK

REVISION_MARK

DRAWING_REVISION_MARK

REV

TITLE

DRAWING_TITLE

UDA:<drawing user-defined attribute>

TPL:<template attribute>

<variable>?- <text>

The question mark (?) can be used for querying if a <variable> is set, and if it does, the <text> after the question mark will be printed. See the second example below.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

Example

```
%DRAWING_NAME.% - %DRAWING_TITLE%%DRAWING_REVISION? - Rev %
%REVISION_MARK%
```

```
%NAME% - %TITLE%%UDA:DRAWING_USERFIELD_1? - %
%UDA:DRAWING_USERFIELD_1%%DRAWING_REVISION? - Rev%
%DRAWING_REVISION%
```

See also

Customize print file names

XS_DRAWING_PLOT_FILE_NAME_C

Category: **Printing**

Use to define print file names for cast unit drawings.

Enter any combination of text and options:

NAME

NAME.-

NAME.

DRAWING_NAME

DRAWING_NAME.

DRAWING_NAME.-

REVISION

DRAWING_REVISION

REV_MARK

REVISION_MARK

DRAWING_REVISION_MARK

REV

TITLE

DRAWING_TITLE

UDA:<drawing user-defined attribute>

TPL:<template attribute>

<variable>?- <text>

The question mark (?) can be used for querying if a <variable> is set, and if it does, the <text> after the question mark will be printed. See the second example below.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

Example

```
%DRAWING_NAME.% - %DRAWING_TITLE%%DRAWING_REVISION? - Rev %
%REVISION_MARK%
```

```
%NAME% - %TITLE%%UDA:DRAWING_USERFIELD_1? - %
%UDA:DRAWING_USERFIELD_1%%DRAWING_REVISION? - Rev%
%DRAWING_REVISION%
```

See also

Customize print file names

XS_DRAWING_POINT_SCALE

Category

Drawing Properties

Use to scale the points Tekla Structures uses to create reference lines. Enter the scale as a decimal. The default value is 0.5.

This advanced option is model specific and the setting is saved in the options database.

XS_DRAWING_RENDERING_ENGINE

Category: Drawing View

Set this advanced option to `FOG` to activate the FOG rendering engine. When set to `FOG`, the overall rendering is faster, which can be seen when you pan or zoom, for example.

To use the Windows Graphics Device Interface (GDI) rendering, set this advanced option to `GDI`. `GDI` is the default value.

Note that setting the XS_USE_SMART_PAN (page 485) advanced option to `TRUE` is useful for the Windows GDI only and has no effect on FOG rendering.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`.

XS_DRAWING_SCALE_SEPARATOR_CHAR

Category

Drawing Properties

Defines the separator character used in drawing scales. The default character is colon (:).

This advanced option is model specific and the setting is saved in the options database.

XS_DRAWING_SHEET_HEIGHT

Category

Drawing View

Use to define the default height of a drawing sheet. The default value is `800`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_DRAWING_SHEET_POSITION_X (page 217)

XS_DRAWING_SHEET_POSITION_X

Category

Drawing View

Use to define the initial position of the drawing sheet. This is useful when using a dual display. The default value is `0`.

You can set this in the following ways:

```
XS_DRAWING_SHEET_POSITION_X=50XS_DRAWING_SHEET_POSITION_Y=50XS_DRAWING_SHE
ET_HEIGHT=600XS_DRAWING_SHEET_WIDTH=900
```

X and Y are the coordinates of the upper left corner of the drawing view, measured from the upper left corner of the MDI client window (the dark gray area in the Tekla Structures window).

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_DRAWING_SHEET_POSITION_Y (page 218)

XS_DRAWING_SHEET_HEIGHT (page 217)

XS_DRAWING_SHEET_WIDTH (page 219)

XS_DRAWING_SHEET_POSITION_Y

Category

Drawing View

Use to define the initial position of the drawing sheet. This is useful when using a dual display. The default value is `0`.

You can set this in the following ways:

```
XS_DRAWING_SHEET_POSITION_X=50 XS_DRAWING_SHEET_POSITION_Y=50
XS_DRAWING_SHEET_HEIGHT=600 XS_DRAWING_SHEET_WIDTH=900
```

X and Y are the coordinates of the upper left corner of the drawing view, measured from the upper left corner of the MDI client window (the dark gray area in the Tekla Structures window).

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_DRAWING_SHEET_POSITION_X (page 217)

XS_DRAWING_SHEET_HEIGHT (page 217)

XS_DRAWING_SHEET_WIDTH (page 219)

XS_DRAWING_SHEET_WIDTH

Category

Drawing view

Use to define the default width of a drawing sheet. The default value is `1000`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_DRAWING_SHEET_POSITION_X (page 217)

XS_DRAWING_SHEET_POSITION_Y (page 218)

XS_DRAWING_SHEET_HEIGHT (page 217)

XS_DRAWING_SINGLE_PART_HATCH_SCHEMA

Category

Hatching

Use to determine the name of the schema file used for single part drawings.

This advanced option is model specific and the setting is saved in the options database.

Example

To use the default schema file, enter `single.htc`.

XS_DRAWING_SNAPSHOT_CREATION

Category

Drawing Properties

If you set the this advanced option to `FALSE`, the snapshots are not created automatically when you save a drawing. The default value is `TRUE`, which means that snapshots are created automatically when you save a drawing.

This advanced option is model specific and the setting is saved in the options database.

XS_DRAWING_SOLID_MERGE_TOLERANCE

Category

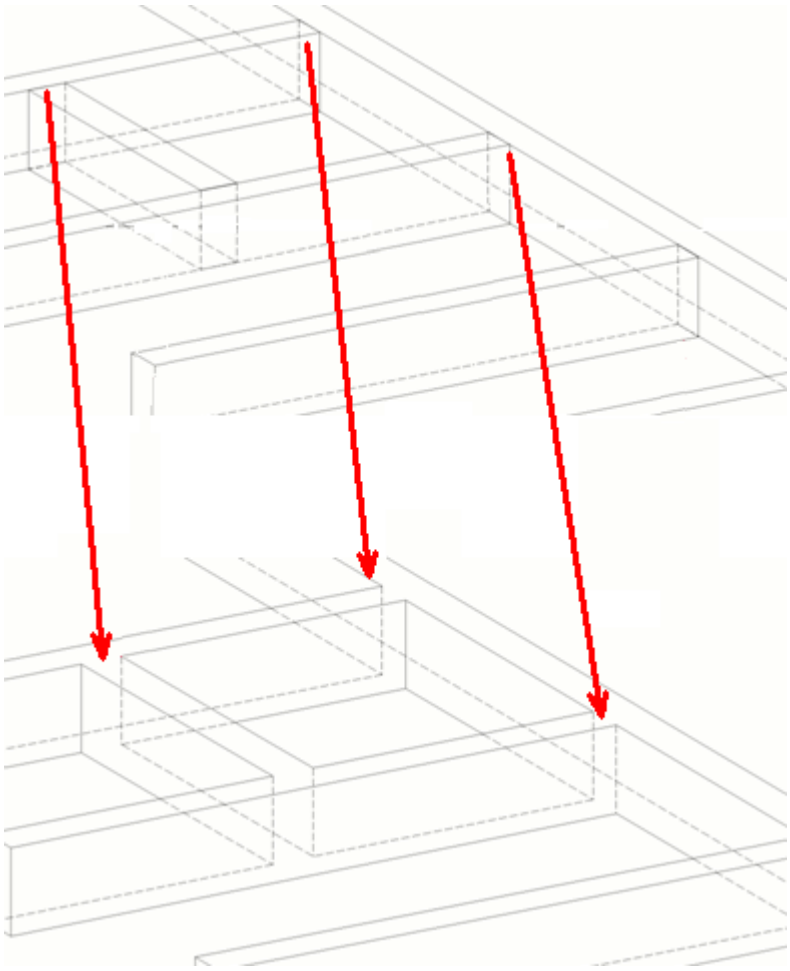
Drawing View

Use to define the limit that determines whether some objects in a cast unit are merged in the drawing view. Enter the value in millimeters. The default value is `6.0`.

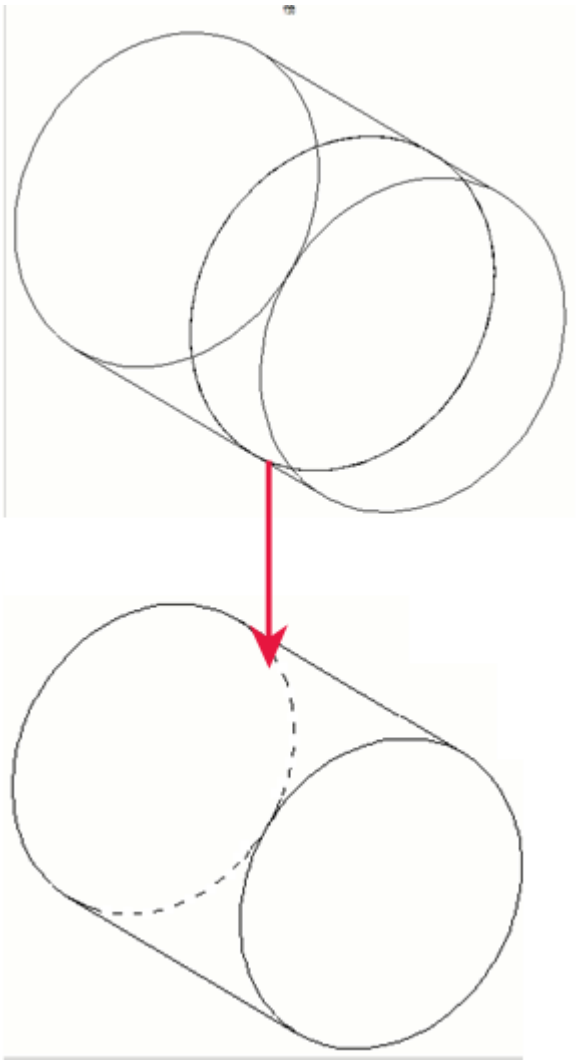
This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

Example

The following example shows the result of merging parts.



In the following example, unnecessary segments of curved parts that are one above another are removed.



XS_DRAWING_STUD_REPRESENTATION

Category

Drawing Properties

Use this advanced option to define different representation options for bolts and studs.

Set the advanced option to `SOLID` (default) to have the studs drawn as solid objects regardless of the bolt representation setting, and to `AS_BOLT` to have the studs drawn according to the settings in the **Bolt properties** dialog box.

This advanced option is role-specific. When you change the value, it changes from system- to model-specific, and its value is the same for all users in the current model.

XS_DRAWING_TEMPLATES_LIBRARY

Category: Drawing Properties

You define the location of the drawing template library by setting this advanced option to point to the model folder that contains the template drawings.

This advanced option is model specific and the setting is saved in the options database. .

Example

`C:\TeklaStructuresModels\CloningTemplate`

(where CloningTemplate is the model name).

To use the a cloning template library and the cloning template:

1. Open the `user.ini` file located in the folder `..\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings` in a text editor.
2. Set the advanced option `XS_DRAWING_TEMPLATES_LIBRARY` to point to the model folder that contains the cloning templates (template library):

   ```
   set XS_DRAWING_TEMPLATES_LIBRARY=%XS_RUNPATH%\DrawingLibrary
   ```

   For example:

   ```
   set XS_DRAWING_TEMPLATES_LIBRARY=C:\TeklaStructuresModels\CloningTemplate
   ```

   (where `CloningTemplate` is the model name).
3. To open the **Clone Drawing** dialog box, click **Clone** in **Document manager**.
4. Use the **Objects and actions in cloning** options to define the drawing objects to be cloned and the actions for each cloned object.
5. Select the **Clone from** > **Other model** option. You can see that the `CloningTemplate` folder is displayed in the box.
6. Click the **Select template...** button.
7. In the **Drawing Templates** dialog box, select the cloning template.
8. Leave the list open and clone the drawing by clicking **Clone selected**.

XS_DRAWING_UDAS_MODIFY_ALL_DRAWING_TYPES

Category: **Drawing Properties**

Use to modify the user-defined attributes of all selected drawings in **Document manager** at the same time, even if the drawings are of different types.

- To allow the modification of user-defined attributes for all drawings types at the same time, set the advanced option to `TRUE`. `TRUE` is the default value.
- To allow the modification of user-defined attributes only for one drawing type at a time, set the advanced option to `FALSE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_DRAWING_UPDATE_VIEW_PLACING

Category

Drawing Properties

Use to control how the view placing works. If the views do not fit on the screen after scaling, Tekla Structures increases the sheet size using the layout properties if autosizing is enabled (**Layout** --> **Drawing size** --> **Size definition mode** --> **Autosize**). Use this advanced option together with XS_INTELLIGENT_DRAWING_ALLOWED (page 287).

This advanced option is model specific and the setting is saved in the options database.

Value	Description
`TRUE`	Updates the view placing. Does not affect the scale or the sheet size.
`TRUE, SCALE`	Updates the view placing and scales the view if the view does not fit on the sheet. Tekla Structures scales the views smaller using the scales in layout properties.
`TRUE, SHEET`	Updates the view and increases the sheet size if the view does not fit on the sheet.
`TRUE, SHEET, SCALE`	Updates the view, scales the view and increases the sheet size, if necessary.
`TRUE, CLONING_ONLY`	Updates the view placing. Does not affect the scale or the sheet size. View placing is updated only during cloning, not during update. `CLONING_ONLY` is the default value.

Value	Description
TRUE, SCALE, CLONING_ONLY	Updates the view placing and scales the view if the view does not fit on the sheet. Tekla Structures scales the views smaller using the scales in layout properties. View placing is updated only during cloning, not during update.
TRUE, SHEET, CLONING_ONLY	Updates the view placing and increases the sheet size if the view does not fit on the sheet. View placing is updated only during cloning, not during update.
TRUE, SHEET, SCALE, CLONING_ONLY	Updates the view placing, scales the view and increases the sheet size, if necessary. View placing is updated only during cloning, not during update.
FALSE	Does not update the view placing or change the view scale or sheet size.

XS_DRAWING_USE_WORKSHOP_FORM _FOR_DOUBLE_PARTS_IN_SINGLE_PART_DRAWINGS

Category

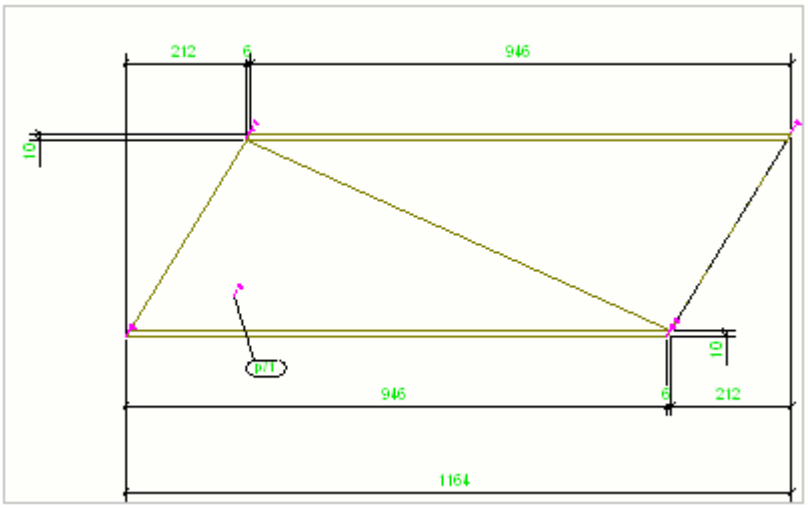
Drawing Properties

Set this advanced option to TRUE to show haunch profiles always in workshop form as double parts in single part drawings. TRUE is the default value. To show the haunch profiles using the part representation selected in the part properties dialog box, set this advanced option to FALSE. This setting affects only part representation, not dimensioning or other part properties.

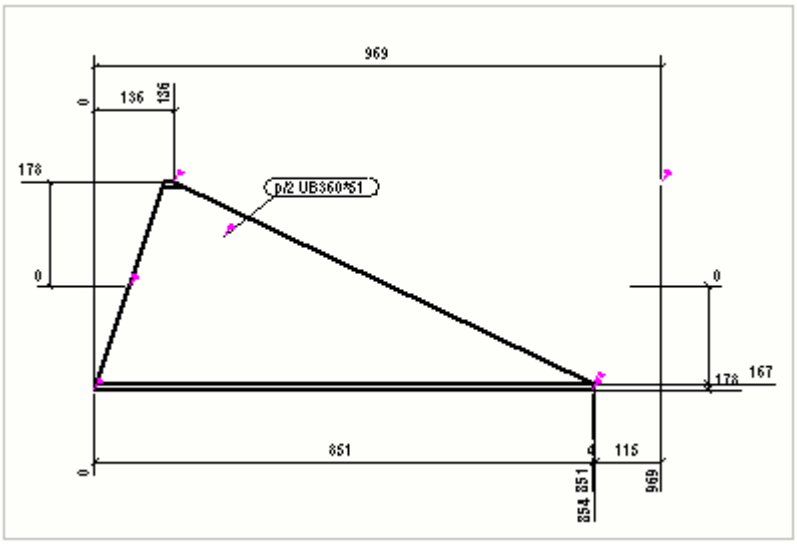
By default, a haunch (translations checked in drawing.ail) has different length calculations and uses different part presentation in drawings than a beam. The calculation routine checks the part name, and when the name "HAUNCH" or translations of it is found, the haunch calculation is used.

You can add additional names for haunch profiles in drawing.ail file under the \messages folder. Use the strings like drawing_haunch_2 or drawing_haunch_3.

Below is an example of the workshop form representation.



Below is an example of the outline representation.



This advanced option is model specific and the setting is saved in the options database.

XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_FRONT

Category

Drawing properties

Use the following advanced options to define the arrow symbol used in section and end view direction marks, for each basic view type (front, top, back, bottom):

- XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_FRONT

- XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_TOP
- XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_BACK
- XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_BOTTOM

By default Tekla Structures uses the symbol no 66 in the xsteel.sym file (located usually in the folder \environments\common\symbols\).

This advanced option is user specific and the setting is saved in options.bin under the user folder, for example, in C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings. Restart Tekla Structures to activate the new value.

See also

Section view properties (page 682)

XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_TOP

Category: Drawing properties

Use the following advanced options to define the arrow symbol used in section and end view direction marks, for each basic view type (front, top, back, bottom):

- XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_FRONT
- XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_TOP
- XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_BACK
- XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_BOTTOM

By default Tekla Structures uses the symbol no 66 in the xsteel.sym file (located usually in the folder \environments\common\symbols\).

This advanced option is user specific and the setting is saved in options.bin under the user folder, for example, in C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings. Restart Tekla Structures to activate the new value.

See also

Section view properties (page 682)

XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_BACK

Category: Drawing properties

Use the following advanced options to define the arrow symbol used in section and end view direction marks, for each basic view type (front, top, back, bottom):

- XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_FRONT
- XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_TOP
- XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_BACK
- XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_BOTTOM

By default Tekla Structures uses the symbol no 66 in the `xsteel.sym` file (located usually in the folder `\environments\common\symbols\`).

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

Section view properties (page 682)

XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_BOTTOM

Category: Drawing properties

Use the following advanced options to define the arrow symbol used in section and end view direction marks, for each basic view type (front, top, back, bottom):

- XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_FRONT
- XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_TOP
- XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_BACK
- XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_BOTTOM

By default Tekla Structures uses the symbol no 66 in the `xsteel.sym` file (located usually in the folder `\environments\common\symbols\`).

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

Section view properties (page 682)

XS_DRAWING_VIEW_REFERENCE_SYMBOL

Category

Drawing Properties

Use to define the symbol that is displayed in drawing views, when you have selected the option **Custom** in the **Symbol** list in the **Detail** or **Section Symbol Properties** dialog box, or in the view properties dialog boxes of other drawing views. For example, if you enter `xsteel@3`, Tekla Structures uses the symbol number 3 in symbol file `xsteel.sym`.

This advanced option is model specific and the setting is saved in the options database.

XS_DRIVER

Category: File Locations

The print device definitions you create in the **Printer Catalog** (**File menu** --> **Printing** --> **Printer catalog**) are located in the `plotdev.bin` file. This file is located in the system folder defined for the advanced option `XS_SYSTEM`.

The definitions in the system folder are accessible to all users. You can also save printer definitions in the current model folder or in the project and firm folders, and in a folder indicated by this advanced option. Tekla Structures searches first for `plotdev.bin` in the model, project and firm folders, then in the folder indicated by the advanced option `XS_DRIVER`.

This advanced option is system specific and is read from environment files. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

XS_DSTV_CREATE_AK_BLOCK_FOR_ALL_PLATES

Category: CNC

Set to `TRUE` (default) to create AK blocks in DSTV files also for rectangular plates, combined NC files and part lists.

When you set the advanced option to `FALSE`, Tekla Structures checks whether an AK block needs to be created for the plate, and creates the AK block when needed.

The DSTV standard does not require the creation of AK blocks if the part is completely described by its length, its dimensions and its skewed cuts in the header data. This advanced option allows you to create AK blocks for plates even if the header data already covers all the information of the plate.

This advanced option is model specific and the setting is saved in the options database.

XS_DSTV_CREATE_AK_BLOCK_FOR_ALL_PROFILES

Category

CNC

When you set the advanced option to `TRUE`, Tekla Structurescreates AK blocks in DSTV files for all profiles, combined NC files and part lists. The default value is `TRUE`.

When you set the advanced option to `FALSE`, Tekla Structures checks whether an AK block needs to be created for the part, and creates the AK block when needed.

The DSTV standard does not require the creation of AK blocks if the profile is completely described by its length, its dimensions and its skewed cuts in the header data. This advanced option allows you to create AK blocks for profiles even if the header data already covers all the information of the profile.

This advanced option is model specific and the setting is saved in the options database.

XS_DSTV_CREATE_NOTCH_ONLY_ON_BEAM_CORNERS

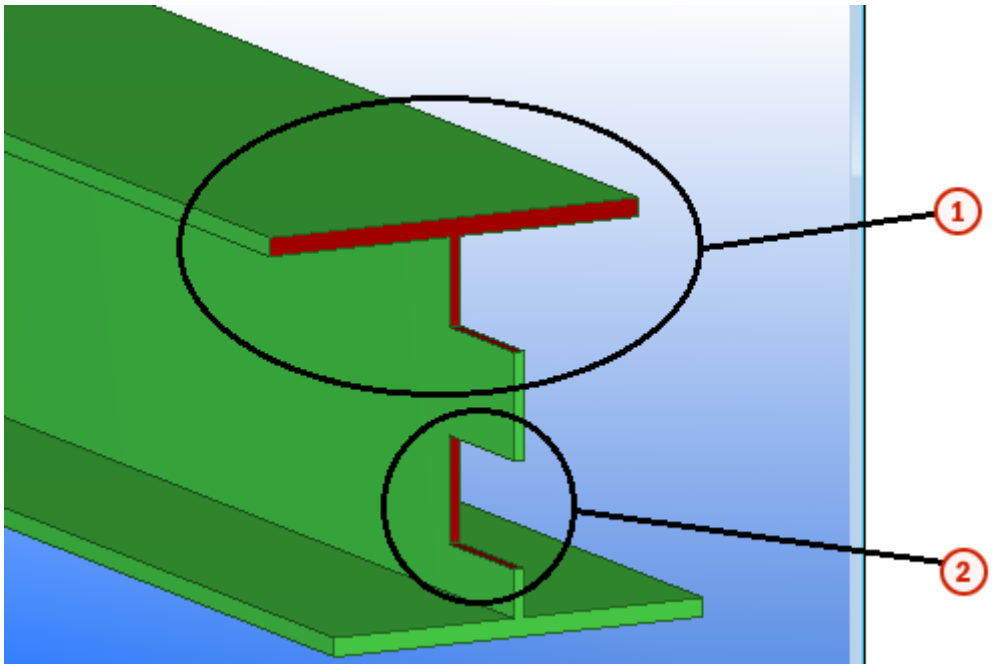
Category: CNC

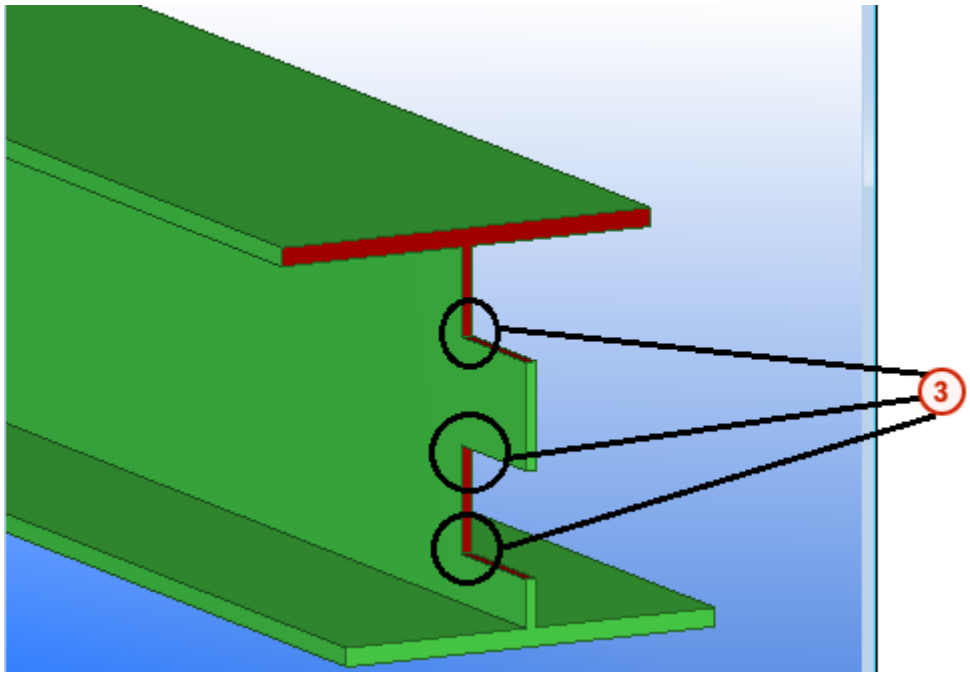
Use this advanced option to control the rounding of notch corners. The default is `FALSE`.

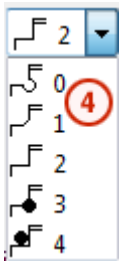
This advanced option is model specific and the setting is saved in the options database.

Examples

The following example describes the concepts of notches, notch corners, notch corner roundings, and beam corners:



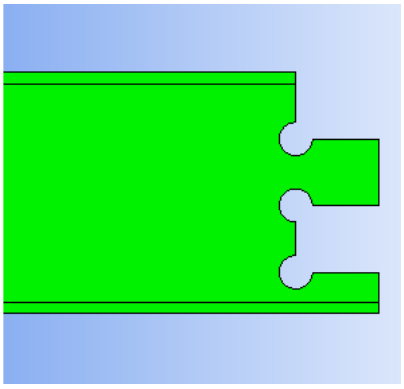
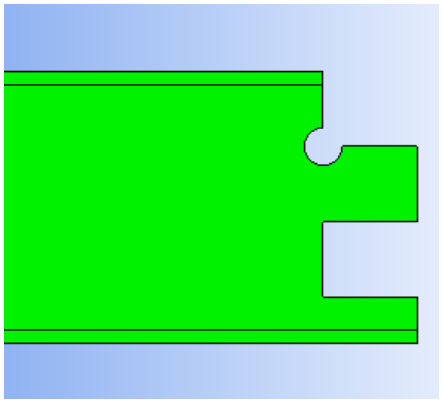
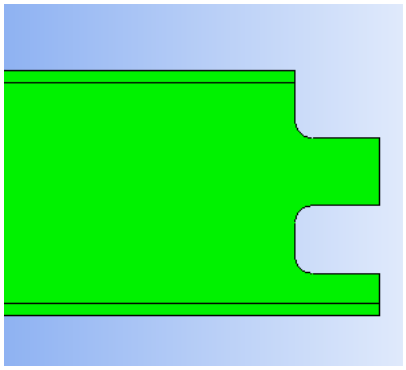
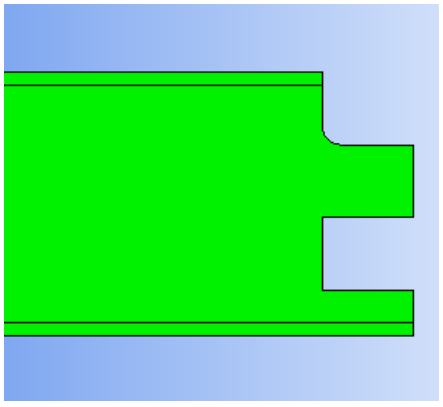




1. The notch is at the beam corner

2. The notch is not at the beam corner
3. Three notches, other corners are ordinary corners
4. Notch corner inner shape (or notch corner rounding) options in the **NC File Settings** dialog box

The table below shows how the `XS_DSTV_CREATE_NOTCH_ONLY_ON_BEAM_CORNERS` setting (`TRUE`/`FALSE`) and the **Inner corners shape** setting affect the NC file.

	`XS_DSTV_CREATE_NOTCH_ONLY_ON_BEAM_CORNERS` set to `FALSE`	`XS_DSTV_CREATE_NOTCH_ONLY_ON_BEAM_CORNERS` set to `TRUE`
Inner corners shape = 0		
Inner corners shape = 1		

XS_DSTV_DO_NOT_UNFOLD_POLYBEAM_PLATES

Category

Category: CNC

Set this advanced option to `TRUE` if you do not want to unfold polybeam plates when you create DSTV files. This means that the polybeam plates will be handled as “cut into shape” instead of “bent into shape” regardless of the

modeling method. For this advanced option to work, the polybeam plate must lie in XY plane of the material.

If set this advanced option to `FALSE`, Tekla Structures writes the unfolded geometry of the polybeam plates to the DSTV files. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_DSTV_LIST_NET_WEIGHT

Category: CNC

When you set `XS_DSTV_LIST_NET_WEIGHT` to `TRUE`, net weight is used in MIS list export. If you set it to `FALSE`, gross weight is used. `FALSE` is the default value.

This advanced option is model specific and the setting is saved in the options database.

XS_DSTV_LIST_SEPARATOR

Category

CNC

Use to define the separator used in DSTV lists. By default the separator is the # character.

This advanced option is model specific and the setting is saved in the options database.

XS_DSTV_NET_LENGTH

Category

CNC

Set this advanced option to `TRUE` to make cuts affect the part length in the NC file header. Set this advanced option to `FALSE` to make only fittings affect the length.

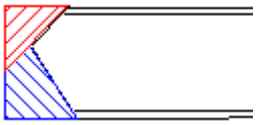
Using this advanced option also affects MIS values, such as KISS and EJE.

This advanced option is model specific and the setting is saved in the options database.

For advanced users

The AK block of the NC file always contains the correct net length. Using this advanced option writes the net length, instead of the length, to the header block. Some NC machines take the length information either from the header or AK-block. Check with the workshop if you are uncertain which method to use.

NOTE Using this advanced option may damage cutting machines if the part contains cuts and fittings and the longest length is not at the edge of the part (machine tries to start cutting in the middle of the part):



See also

XS_DSTV_PRINT_NET_AND_GROSS_LENGTH (page 236)

XS_DSTV_NO_SAWING_ANGLES_FOR_PLATES_NEEDED

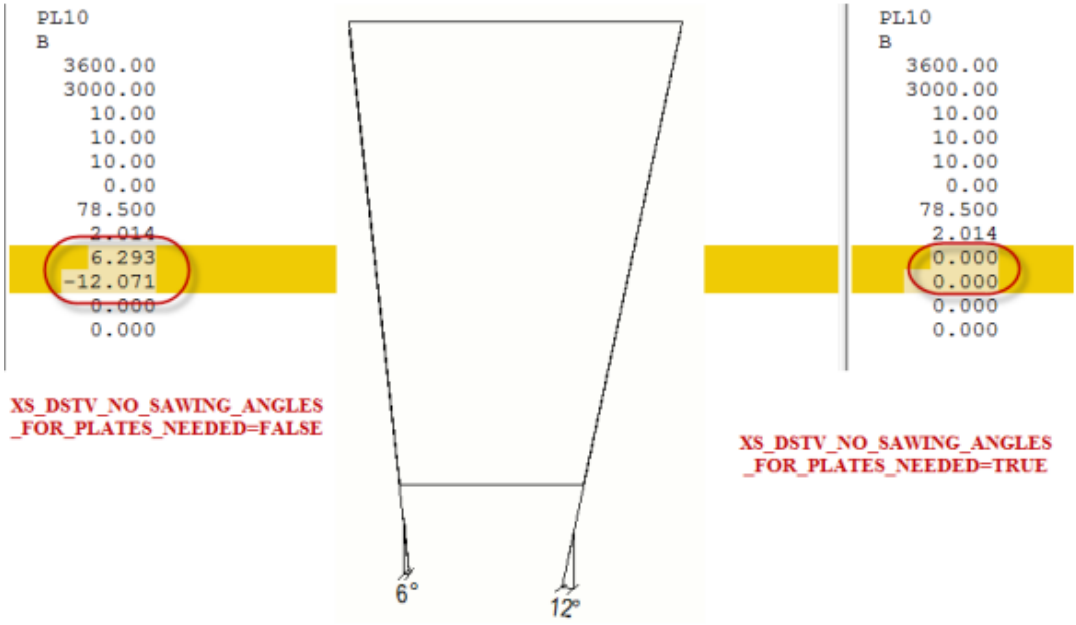
Category

CNC

With this variable you can define whether you need to specify skew angles in NC file header or not for plates. `TRUE` does not write the skew angles in the file heading. If you want to specify skew angles, set this advanced option to `FALSE`. The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

See below for an example:



XS_DSTV_NUMBER_OF_PARTS_BY_SELECTION

Category

CNC

Use to add the number of parts in the header of an NC file by the parts that have been selected in the model.

When you set this advanced option to `TRUE`, and select the **Create for selected parts** option in the **NC Files** dialog box, the number of parts in the header of the NC file matches the number of selected parts.

The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_DSTV_PLATE_PROFILE_WITH_WIDTH

Category

CNC

Set this advanced option to `TRUE` to have both the plate thickness and the plate profile width written in the DSTV file header. Set this advanced option to

FALSE to have only the plate thickness written in the DSTV file header. The default value is FALSE.

This advanced option is model specific and the setting is saved in the options database.

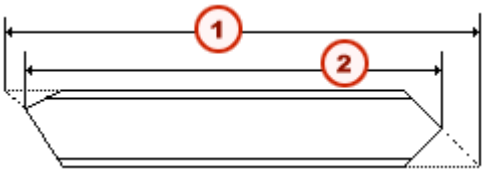
XS_DSTV_PRINT_NET_AND_GROSS_LENGTH

Category

CNC

Set this advanced option to TRUE to insert two length values in NC DSTV files:

- Gross length
- Net length

If you do not want to do this, set this advanced option to FALSE. FALSE is the default value.

1 Gross length

2 Net length

This advanced option is model specific and the setting is saved in the options database.

NOTE Net and gross length may switch places in the NC file if the advanced option XS_CHECK_FLAT_LENGTH_ALSO is set. Then Tekla Structures may use the length value it finds in the fltprops.inp file instead.

See also

XS_DSTV_NET_LENGTH (page 233)

Fittings and line cuts in NC files

XS_CHECK_FLAT_LENGTH_ALSO (page 117)

XS_DSTV_REAL_WIDTH_INTO_HEADER_PROFILE_FOR_PLATES

Category

CNC

Set to `TRUE` to have the real plate width instead of the nominal plate width in the header information of DSTV files. The default is `FALSE`.

Here is an example of the difference between the real plate width and the nominal plate width: A user models a plate beam using the profile `PL200*10`, but then uses part or line cuts to create a 5 mm cut along the length of the plate, for example, to create a root opening for a weld, so that the end result is a plate which is only 195 mm wide. In this case, the real plate width would be 195 mm, and the nominal plate width 200 mm.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DSTV_USE_COUNTERSUNK_HOLES

Category: CNC

If you set `XS_DSTV_USE_COUNTERSUNK_HOLES` to `FALSE` in **File menu** --> **Settings** --> **Advanced options** --> **CNC**, Tekla Structures will not create CSK holes in DSTV export. The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

XS_DSTV_USE_EQUAL_ACCURACY_FOR_PLATE_PROFILE_AND_WIDTH

Category

CNC

Set this advanced option to `TRUE` if you want the plate profile width value, plate width value in header and y coordinate values in AK and IK blocks to be rounded to the nearest millimeter. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_DSTV_USE_ONE_VERTEX_SHARP_INNER_CORNER

Category

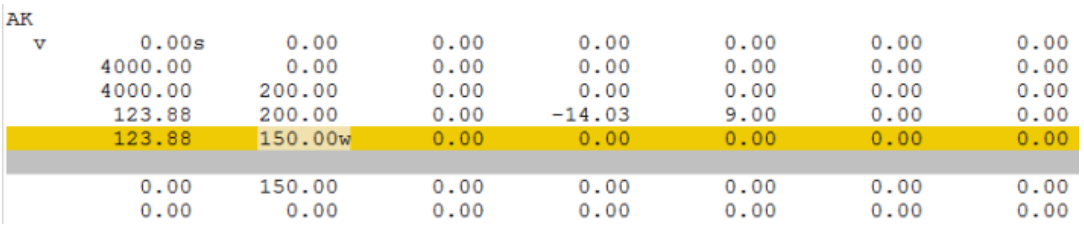
CNC

Set this advanced option to `TRUE` if you do not need to add additional vertex points in the AK block of your DSTV file, for example, when you do not define the rounding radius in the NC settings. By default, this advanced option is set to `FALSE`, which means that additional vertex points are added.

This advanced option is model specific and the setting is saved in the options database.

Example

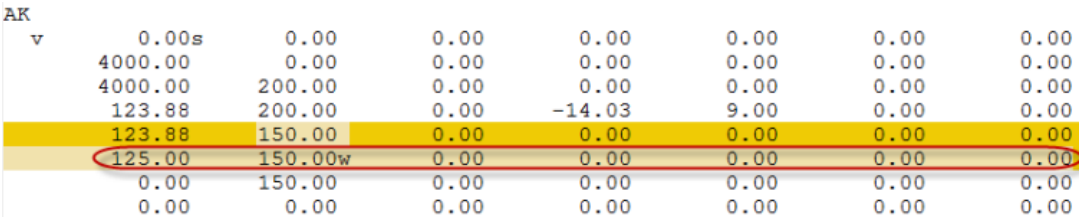
Results in the following DSTV file when the advanced option is set to `TRUE`:



```
AK
  v       0.00s       0.00       0.00       0.00       0.00       0.00       0.00
       4000.00       0.00       0.00       0.00       0.00       0.00       0.00
       4000.00     200.00       0.00       0.00       0.00       0.00       0.00
        123.88     200.00       0.00     -14.03       9.00       0.00       0.00
        123.88     150.00w      0.00       0.00       0.00       0.00       0.00

          0.00     150.00       0.00       0.00       0.00       0.00       0.00
          0.00       0.00       0.00       0.00       0.00       0.00       0.00
```

Results in the following DSTV file when the advanced option is set to `FALSE`.



```
AK
  v       0.00s       0.00       0.00       0.00       0.00       0.00       0.00
       4000.00       0.00       0.00       0.00       0.00       0.00       0.00
       4000.00     200.00       0.00       0.00       0.00       0.00       0.00
        123.88     200.00       0.00     -14.03       9.00       0.00       0.00
        123.88     150.00       0.00       0.00       0.00       0.00       0.00
        125.00     150.00w      0.00       0.00       0.00       0.00       0.00
          0.00     150.00       0.00       0.00       0.00       0.00       0.00
          0.00       0.00       0.00       0.00       0.00       0.00       0.00
```

XS_DSTV_USE_REAL_DIMENSIONS_IN_HEADER

Category

CNC

Set to `TRUE` to have bounding box values of the height and width written to the profile header data in the NC file. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DSTV_WRITE_BEHIND_FACE_FOR_PLATE

Category

CNC

Set to `TRUE` to write (AK + IK) contours for front (v) and back (h) faces of plates in NC DSTV files. Set to `FALSE` to Tekla Structures only write the front face for plate profiles. The default value is `FALSE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_DUPLICATE_CHECK_LIMIT_FOR_COPY_AND_MOVE

Category

Modeling Properties

Use to define the maximum number of objects that are checked for duplicates while copying or moving objects.

If the selection contains too many objects, Tekla Structures does not check for duplicates. Enter an integer value. The default value is `100`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_DWG_EXPORT_UPDATE_TS_LINEWORK_OPTION

Category: Export

The setting **Update Tekla Structures linework only** is only shown in the **Export drawings to DWG/DXF** dialog box if you have set the new advanced option `XS_DWG_EXPORT_UPDATE_TS_LINEWORK_OPTION` to `TRUE`. **Update Tekla Structures linework only** updates the Tekla Structures drawing content only and keeps other content that is created in a CAD software intact at the same file. Blocks (groups), created by Tekla Structures will be updated. Note that you need to have the same drawing exported already, and the layer setup and the layer template must be the same as during the previous export. All CAD lines that were added previously will stay in the file and only Tekla

Structures content will get updated, unless editing was done in CAD blocks editor. The default value is `FALSE`.

This setting is user specific, and it is saved in the `options.bin` under the user folder.

Note that if you edit content of a block (CAD object), and then select the **Update Tekla Structures linework only** option, the whole block will be re-written, and the changes made in CAD will not be kept. To keep the changes in CAD, you need to explode a block before editing it.

For example, you may want to use this option if you have added drawing title blocks in the CAD after the first export of the drawing from Tekla Structures, and want to keep these tables as they are, and update only the objects that exported from Tekla Structures.

For more information about the DWG export, see .

XS_DWG_IMPORT_IGNORE_UNITS

Category

This advanced option must be set in an initialization (.ini) file.

If a DWG reference file is created with imperial settings, it will import to Tekla Structures in too large scale. You can use this advanced option to prevent this.

If you want all coordinates to be meters, set this advanced option to `TRUE`.If you want to take the unit from the DWG file based on the measurement and $insunit definitions in the file header, leave the value out. By default, this advanced option is not set to any value.

XS_DXF_FONT_CONVERSION_FILE

Category

Printing

Use this advanced option to specify a font conversion file for Tekla Structures drawings' DWG and DXF export and printing. The font conversion file defines which font file will be connected to the created AutoCAD style, and optional font width and height correction factors. You can use both True Type font files and AutoCAD `.SHX` font files.

If only the file name of the font conversion file is given, the file is read from the model folder. To use a font conversion file from another location, give the file name with relative or full path. If `XS_DXF_FONT_CONVERSION_FILE` is not set, Tekla Structures will try to use the default font conversion file (`dxf_fonts.cnv`) from the folder defined by `DXK_FONTPATH`. `DXK_FONTPATH` is defined in `teklastructures.ini`.

If the font conversion file is not found or if it does not contain a mapping for a certain font, the font name used in Tekla Structures is used to form the name of the text style in AutoCAD. Spaces are replaced with underscores and lowercase letters are replaced with uppercase ones. For example, font name Arial Narrow in Tekla Structures will become a style named ARIAL_NARROW in AutoCAD.

In addition to the font-specific width and height correction factors defined in the font conversion file, there are general variables `XS_DXF_TEXT_HEIGHT_FACTOR` and `XS_DXF_TEXT_WIDTH_FACTOR`, which apply to all exported texts regardless of the font. If both font specific factors and general variables are used, they are multiplied.

Syntax used in font mapping:

```
Font name in Tekla Structures = Font file name in AutoCAD [*
width correction factor [* height correction factor]]
```

Examples of font mappings in a `.cnv` file:

```
Arial Narrow = ARIALN.TTF
Arial Narrow Bold Italic = ARIALNBI.TTF * 0.5 * 1.0
```

NOTE
- The names in the font conversion file are case sensitive.
- The font conversion file is used for drawings in DWG and DXF export and printing only, it does not affect drawings DWG and DXF import or model import or export.

See also

DXK_FONTPATH (page 242)

XS_DXF_TEXT_HEIGHT_FACTOR (page 241)

XS_DXF_TEXT_WIDTH_FACTOR (page 242)

XS_DXF_FONT_NAME

Category

Printing

Use to specify a font other than the default for 2D DXF files. By default, this advanced option is not set to any value.

This advanced option is model specific and the setting is saved in the options database.

XS_DXF_TEXT_HEIGHT_FACTOR

Category

Printing

Use to set a scale factor for 2D DXF text height. Enter the factor as a decimal. The default value is `1.0`.

This advanced option is model specific and the setting is saved in the options database.

XS_DXF_TEXT_WIDTH_FACTOR

Category

Printing

Use to set the scale factor for 2D-DXF text width. Enter the factor as a decimal. The default value is `1.0`.

This advanced option is model specific and the setting is saved in the options database.

DXK_FONTPATH

Category

This advanced option must be set in an initialization (.ini) file.

This advanced option is system specific and is read from `teklastructures.ini`. It can also be set locally, see your environment `.ini` file (`env_<environment_name>.ini`). Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

Use to point to the folder containing Tekla Structures graphic fonts. The Template editor, for example, uses graphic fonts. `DXK_FONTPATH` is defined in `teklastructures.ini`.

Always end the path with the backslash character.

Example

```
set DXK_FONTPATH=%XSDATADIR%\environments\common\fonts\
```

See also

DXK_SYMBOLPATH

This advanced option must be set in an initialization (.ini) file.

This advanced option is system specific and is read from `teklastructures.ini`. It can also be set locally, see your environment `.ini` file (`env_<environment_name>.ini`). Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

This advanced option points to one or several folders containing Tekla Structures symbol libraries. These folders also contain the DWG files used in snap symbols and handles. The order of folders in `DXK_SYMBOLPATH` is significant: If there are duplicate files, the first one found is used. All files are read from all defined folders.

The `DXK_SYMBOLPATH` is defined in the environment initialization file `env_<environment_name>.ini`, located in `..\Trimble\Tekla Structures\<version>\<environments>\<your_environment\`, and in the Tekla Structures initialization file `teklastructures.ini` located in the `..\Tekla Structures\<version>\bin\` folder.

Separate different folder paths with a semicolon (;). Always end a folder path with a backslash character.

Example

Example with one folder:

```
DXK_SYMBOLPATH=C:\ProgramData\Trimble\Tekla Structures
\<version>\environments\common\symbols\
```

Example with several folders:

```
DXK_SYMBOLPATH=%XS_FIRM%;%XSDATADIR%\environments\uk\General
\symbols\;%XSDATADIR%\environments\common\symbols\
```

In the latter example, Tekla Structures first checks your own symbol files in your firm folder, then the symbol files in the UK environment symbols folder, and last the symbol files in the common environment symbols folder. If a duplicate file exists, the one that Tekla Structures finds first is used.

Using a firm folder for images and symbols

You can define a firm folder where Tekla Structures always searches for the images and symbols. When you store the images and symbols in this folder, you do not have to move them from folder to folder when you install a new version of Tekla Structures. Installing a new version does not replace the files in the firm folder. For more information about defining the firm folder for images and symbols, see .

1.5 Advanced options - E

XS_ENABLE_FAST_CUSTOM_PROPERTY_LOADING

This advanced option must be set in startup .ini files

To switch off the functionality of loading custom properties from the extension folder `..common\extensions\custom\properties\`, set the advanced option `XS_ENABLE_FAST_CUSTOM_PROPERTY_LOADING` to `FALSE` in a startup `.ini` file using the following command:

```
set XS_ENABLE_FAST_CUSTOM_PROPERTY_LOADING=FALSE
```

When you do this, the custom properties are loaded from all folders and subfolders in `..common\extensions`, and from the locations defined in `XS_EXTENSION_DIRECTORY`. This may cause performance issues and loading failures when custom properties are used.

Loading custom properties from `..common\extensions\custom\properties\` speeds up the loading process and solves loading problems caused by other software components in other extension folders.

XS_ENABLE_INNER_CONTOURS_IN_CUT_PARTS

Category

Speed and Accuracy

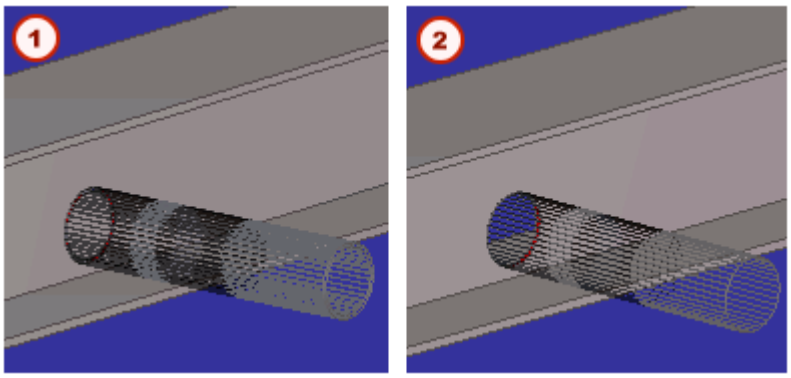
Set this advanced option to `TRUE` to have Tekla Structures create a part cut according to the inner and outer surfaces of the cutting part.

When the advanced option is set to `FALSE`, Tekla Structures creates the cut according to the outer surface of the cutting part. This is the default.

This advanced option is model specific and the setting is saved in the options database.

Example

Here a beam has been cut with a round tube.



1. Advanced option set to `TRUE`
2. Advanced option set to `FALSE`

XS_ENABLE_MIDDLE_BUTTON_DOUBLE_CLICK_ZOOM_ORIGINAL

Category

This advanced option must be set in an initialization (.ini) file.

When you set this advanced option to `TRUE` and double-click the middle mouse button, Tekla Structures zooms the open drawing to its original size.

Example

```
XS_ENABLE_MIDDLE_BUTTON_DOUBLE_CLICK_ZOOM_ORIGINAL=TRUE
```

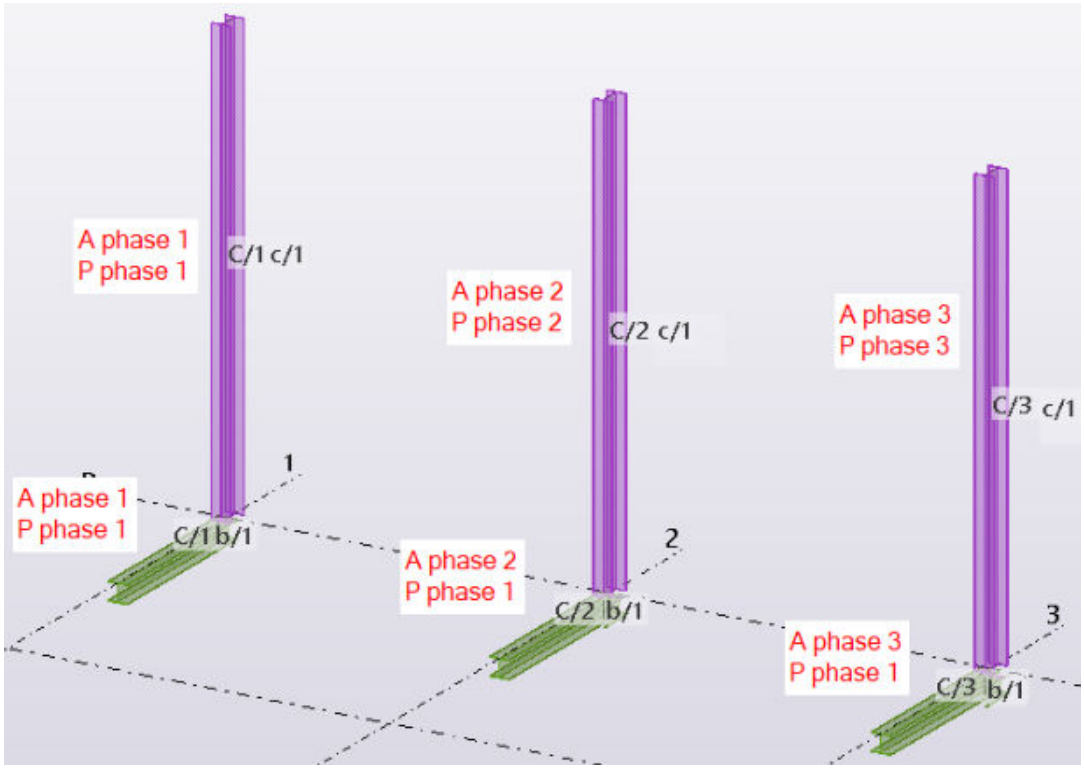
XS_ENABLE_PHASE_OPTION_IN_NUMBERING

Category: Numbering

When this advanced option is set to `FALSE` (which is the default value), the **Assembly phase** check box is disabled in the **Numbering Setup** dialog box. This means that changing an object's phase does not affect numbering or trigger change alerts.

If this advanced option is set to `TRUE`, the **Assembly phase** check box is enabled in the **Numbering Setup** dialog box. This is the recommended setting for Tekla Model Sharing users.

If you select the **Assembly phase** check box, only the assembly phase of each assembly is compared in numbering. This means that otherwise identical objects with different assembly phase get the same part position number.



Note! Even if you **do not** select the **Assembly phase** check box, but you change the phase of an object when `XS_ENABLE_PHASE_OPTION_IN_NUMBERING` is set to `TRUE`:

- A question mark (?) is added to the object's position number indicating that numbering is needed.
- After renumbering, the **Document manager** indicates that the drawings need updating.
- In shared models, the **Changes** list shows the changed parts and assemblies.

This happens even if you are using the Project Viewer configuration.

This advanced option is model specific and the setting is saved in the options database.

See also

General numbering settings (page 629)

XS_ENABLE_POUR_MANAGEMENT

Category: Concrete Detailing

Set this advanced option to `TRUE` to enable pour management in the currently open model and to show cast-in-place concrete structures as continuous. The

commands that show and create pour objects and pour breaks in the model and drawings are only available when pour management is enabled.

The default value for new models in the **Concrete contractor** role is `TRUE`. In the other standard roles, the default value for new models is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

WARNING If pour management is enabled in the model, do not disable it using `XS_ENABLE_POUR_MANAGEMENT`, especially in the middle of the project. This may cause problems if you have drawings containing pour objects, and if you are sharing your model. The pour objects and pour breaks in the model and in the drawings may get invalid, and you may lose pour-related modeling work.

XS_ENABLE_PRECAST_CONTINUOUS_CONCRETE

This advanced option must be set in an initialization (.ini) file.

Set this advanced option to `TRUE` to show precast concrete structures as continuous in model views. Showing precast concrete structures as continuous means that, within each cast unit, the concrete parts with the same material grade and the cast unit type **Precast** are shown as merged so that the part outlines between touching or overlapping parts are not visible.

Note that you also need to set XS_ENABLE_POUR_MANAGEMENT (page 246) to `TRUE` in the **Advanced Options** dialog box.

The default value of `XS_ENABLE_PRECAST_CONTINUOUS_CONCRETE` is `FALSE`, which means that the outlines of precast concrete parts within each cast unit are visible.

Restart Tekla Structures after changing the value to activate the new setting.

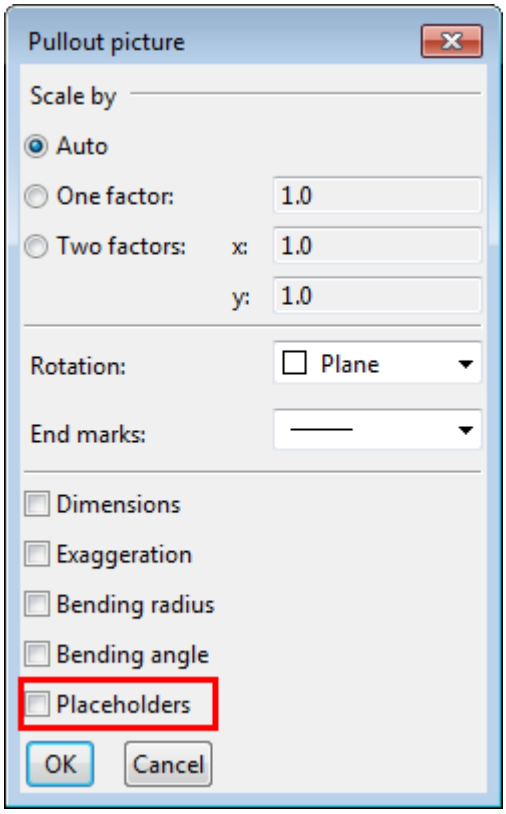
When `XS_ENABLE_PRECAST_CONTINUOUS_CONCRETE` is set to `TRUE`, you can adjust how the precast concrete structures appear in each model view by using the following display settings of cast-in-place concrete structures in **View Properties** --> **Display**:

- Ensure that **Cast in place** is set to **Parts**.
- In the **Cast in place parts** list, select either **Merged** or **Separated** to hide or show the touching or overlapping part outlines.

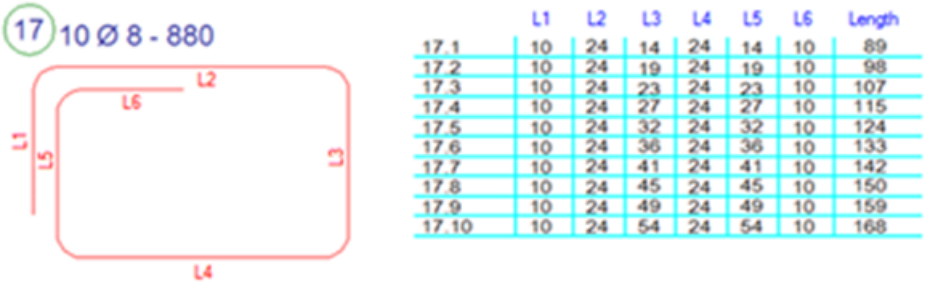
XS_ENABLE_PULLOUT_PLACEHOLDERS

Category: Concrete Detailing

Set this advanced option to `TRUE` to add placeholders instead of dimensions in pull-out pictures. This advanced option adds the option **Placeholders** in the **Pullout picture** dialog box.



The following example shows how the placeholders are used:



	L1	L2	L3	L4	L5	L6	Length
17.1	10	24	14	24	14	10	89
17.2	10	24	19	24	19	10	98
17.3	10	24	23	24	23	10	107
17.4	10	24	27	24	27	10	115
17.5	10	24	32	24	32	10	124
17.6	10	24	36	24	36	10	133
17.7	10	24	41	24	41	10	142
17.8	10	24	45	24	45	10	150
17.9	10	24	49	24	49	10	159
17.10	10	24	54	24	54	10	168

This advanced option is model specific and the setting is saved in the options database.

XS_ENTER_FINALIZES_COMMANDS

Category: **Modeling Properties**

Use this advanced option to set the **Enter** key as a shortcut for completing commands. This advanced option applies also to the direct modification commands.

By default, the advanced option is set to `TRUE`.

XS_ENABLE_OVERLAPPING_CUT_LINE_REMOVAL

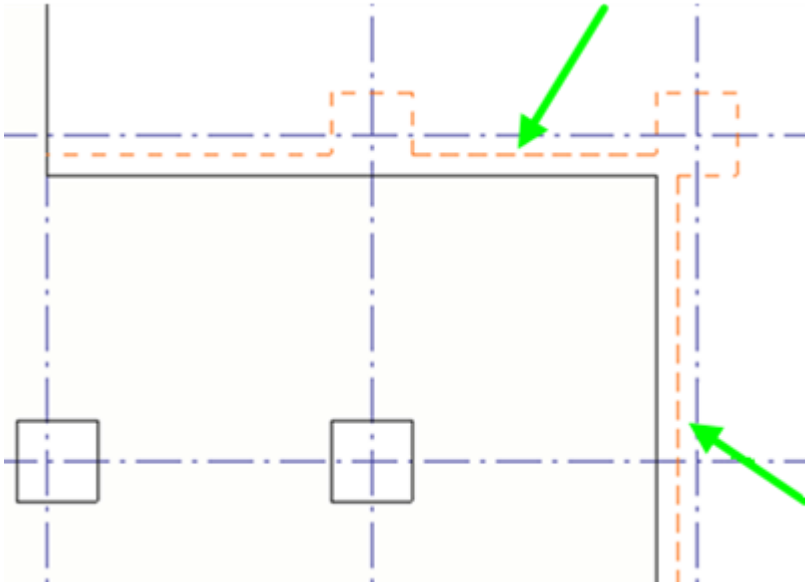
Category: Drawing Properties

This advanced option controls the removal of overlapping cut or non-solid lines. It is by default set to `TRUE`, and the overlapping cut and non-solid (dashed, dash-and-dot) lines are removed.

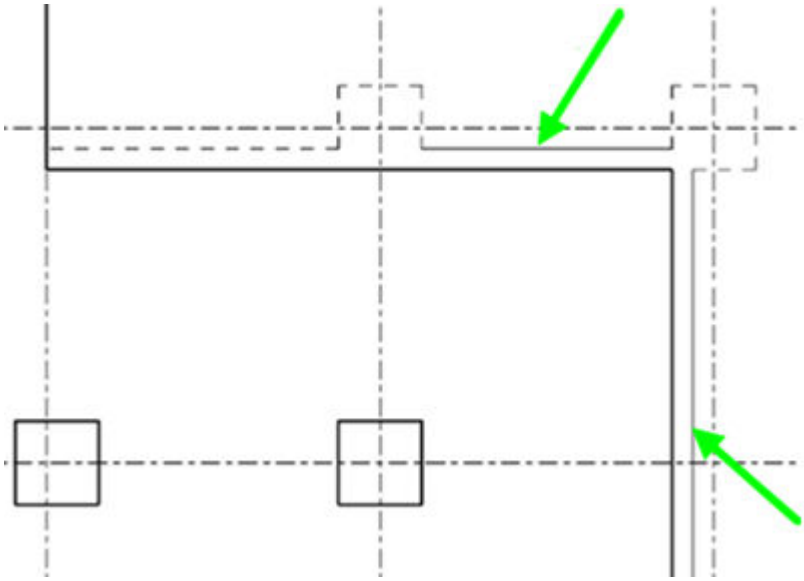
Do not set this advanced option to `FALSE`. If you do that, the cut and non-solid lines may become misaligned and look like solid lines in drawings, and hidden objects may be presented as not hidden, for example.

Note that this advanced option does not remove cut or non-solid lines that are added manually to the drawing.

`XS_ENABLE_OVERLAPPING_CUT_LINE_REMOVAL` set to `TRUE`:



`XS_ENABLE_OVERLAPPING_CUT_LINE_REMOVAL` set to `FALSE`:



This advanced option is model specific and the setting is saved in the options database.

XS_ENABLE_REBAR_MARK_LEADER_LINE_BASE_POINT_OPTIMIZATION

Category: **Concrete Detailing**

Use to select an optimal place for the associativity point of the reinforcing bar mark leader line. If set to `TRUE`, Tekla Structures selects an optimal place for the associativity point. The associativity point only points to one reinforcing bar. The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

XS_EQUAL_SHAPE_DIMENSIONS_TO_BOTH_ENDS_LIMIT

Category

Dimensioning: Parts

Shape dimensions are automatically shown on both ends of a beam, even if the dimensions are the same. You can change this by entering a value in millimeters for this advanced option.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is

in use, you can change the value, which is then the same for all users in the current model.

Example

If you set this advanced option to 300, and a part is 300 mm shorter in one direction, Tekla Structures only displays the dimension in the longer direction. Use millimeters in all environments.

XS_ERASE_UDA_VALUE_WITH_ATTRIBUTE_IMPORT_NULL_AND_BLANK

Category

This advanced option must be set in an initialization (.ini) file.

Set to `TRUE` to erase the values of selected user-defined attributes in attribute import.

To select which values of user-defined attributes are erased, enter any of the following values at the end of the file to be imported, in the same order as the user-defined attributes are entered in the file:

- `NULL`
- `null`
- No value (two consecutive delimiters)

Example

If the content of the input file is the following:

```
ID; USER_FIELD_1; USER_FIELD_2; USER_FIELD_3; USER_FIELD_4;
12345;NULL;null;;4
```

The result is:

User-defined attribute values for 1-3 are erased in the attribute import. User-defined attribute 4 has the value 4 in the attribute import.

NOTE Space and tab are not recommended as delimiters in the input file when this functionality is used.

XS_EXCLUDED_PARTS_IN_ORIENTATIONAL_NUMBERING

Category: Numbering

The advanced option `XS_EXCLUDED_PARTS_IN_ORIENTATIONAL_NUMBERING` can be used in conjunction with the orientation numbering setting. Similar

parts will be numbered the same even if their orientation is different and the orientation setting has been selected in the **Numbering Setup** dialog box. You can enter the desired part names separated by spaces. Wildcards are also allowed. This advanced option is model specific and the setting is saved in the options database.

XS_EXPORT_BREP_AS_EXACT_SOLID

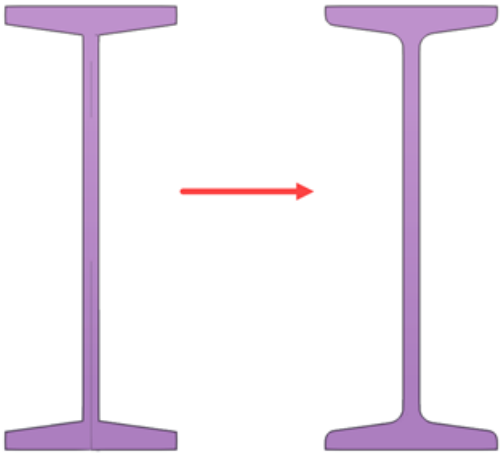
Category: Export

Set this advanced option to `TRUE` to export B-rep objects as exact solids in the IFC2x3 export. `FALSE` is the default value.

Note that if you export B-reps as exact solids, the IFC file size increases, and the export takes more time.

TIP To get smoother edges to the export, you may need to set the advanced option XS_CS_CHAMFER_DIVIDE_ANGLE (page 252) to `10`.

In the following example, on the left, you can see a native type I profile, and on the right the IFC object surface geometry when both advanced options are used:



This advanced option is model specific and the setting is saved in the options database.

See also

XS_CS_CHAMFER_DIVIDE_ANGLE (page 139)

XS_EXPORT_CODEPAGE

Category

Export

Tekla Structures sets the codepage automatically so that exported files are displayed correctly. If the proper codepage cannot be found, the codepage is set to `ansi_1252` by default. You can set the codepage manually by using this advanced option, which overrides the automatic selection of codepage in export. By default, this advanced option is not set to any value.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

To set the advanced option to the required codepage, use one of the following values:

- `ascii`
- `iso8859-1`
- `iso8859-2`
- `iso8859-3`
- `iso8859-4`
- `iso8859-5`
- `iso8859-6`
- `iso8859-7`
- `iso8859-8`
- `iso8859-9`
- `dos437`
- `dos850`
- `dos852`
- `dos855`
- `dos857`
- `dos860`
- `dos861`
- `dos863`
- `dos864`
- `dos865`
- `dos869`

- dos932
- mac-roman
- big5
- ksc5601
- johab
- dos866
- ansi_1250
- ansi_1251
- ansi_1252
- gb2312
- ansi_1253
- ansi_1254
- ansi_1255
- ansi_1256
- ansi_1257
- ansi_874
- ansi_932
- ansi_936
- ansi_949
- ansi_950
- ansi_1361
- ansi_1200
- ansi_1258

XS_EXPORT_DGN_COORDINATE_SCALE

Category

Export

Use this advanced option to set the coordinate scale to use in DGN exports.

DGN coordinate scale does not actually scale the model, but changes the model precision. If you set this advanced option to 100, the accuracy is 1/100 mm.

This advanced option is model specific and the setting is saved in the options database.

XS_EXPORT_DGN_FILENAME

Category

This advanced option must be set in an initialization (.ini) file.

This advanced option has been removed.

Use to indicate the output file name in DGN exports. The default is `model.dgn`.

XS_EXPORT_DGN_INCLUDE_CUTS

Category

Export

Use this advanced option to define the cuts to include in DGN exports. You can use the following values:

Value	Use to
`FALSE`	Exclude all cuts.
`TRUE`	Include all cuts (default value).
`CLASH`	Include all cuts but skip hole cut ends.
`CLASH_NORMAL_PLATES`	Same as `TRUE` for contour plates and `CLASH` for all other parts.

This advanced option is model specific and the setting is saved in the options database.

XS_EXPORT_DGN_INCLUDE_INNER_CONTOUR

Category

Export

Use to include or exclude inner contours of tubes in DGN exports. The options are:

- `TRUE` to include inner contours (default)

- FALSE (default) to exclude inner contours

This advanced option is model specific and the setting is saved in the options database.

XS_EXPORT_DGN_ROUND_SEGMENTS

Category

Export

Use to define the number of segments Tekla Structures uses to display round tubes. Tekla Structures uses this value for large tubes (larger than 100 mm) and 80% of this value for small tubes. The default value is 40.

This advanced option is model specific and the setting is saved in the options database.

XS_EXPORT_DGN_USE_CLASS_AS_COLOR

Category

Export

Set the color for exported parts by the part class (like **Color by class** in the model).

By default the export uses the current colors of the Tekla Structures view (FALSE). Set this advanced option to TRUE when you have defined some other color setting than **Color by class** in the **Object Representation** dialog box, but still want to export with **Color by class**.

This advanced option is model specific and the setting is saved in the options database.

XS_EXPORT_DGN_USE_VOLUMETRIC

Category

Export

Set this advanced option to TRUE to define plates with profile type plate or polygon plate to DGN type attribute 92 (0x05C in DGN cell header) and all other beams to DGN type attribute 91 (0x05B in DGN cell header) in DGN export. If you set it to FALSE, all beams get 91 as the type attribute in DGN export. The default value is FALSE.

When this advanced option is set to `TRUE`, mapping to Microstation is successful and the attribute report in Tekla Structures is correct.

This advanced option is model specific and the setting is saved in the options database.

XS_EXPORT_DRAWING_TRY_TO_KEEP_LOCATION

If you set the advanced option `XS_EXPORT_DRAWING_TRY_TO_KEEP_LOCATION` to `TRUE` (default), Tekla Structures tries to keep the DWG origin in the same position in the export as the drawing view origin. This can only be done in plan views and elevation views. If the drawing has more than one plan view or elevation view, Tekla Structures places the DWG origin in the bottom-left corner of the drawing frame.

If you set this advanced option to `FALSE`, the origin (0,0) is set to the bottom-left corner of the drawing frame.

This advanced option is system specific and is read from `teklastructures.ini`. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

For more information about the drawing export, see .

XS_EXPORT_FILLMODE

Category

Export

Use to control how fills are exported to DWG and DXF formats.

This advanced option is model specific and the setting is saved in the options database.

Value	Description	Appearance in exported DWG/DXF	Appearance in Tekla Structures
`HATCH`	Fills are drawn as hatches in the exported DWG/DXF. This is the default value.		

Value	Description	Appearance in exported DWG/DXF	Appearance in Tekla Structures
BORDER	Only the border lines of fills are drawn in the exported DWG/DXF.		
FILL	Fills are drawn as filled triangles in the exported DWG/DXF.		
NONE	Fills are not drawn in the exported DWG/DXF.		

XS_EXPORT_IFC_REBARSET_INDIVIDUAL_BARS

Category: Export

Use this advanced option to control how rebars created by rebar sets are exported in the IFC2x3 export. To export the bars in groups, set this advanced option to FALSE (default), and to export the bars as individual bars, set it to TRUE.

NOTE In the IFC file, 'Total number' always shows 1 for groups created by rebar sets and 'Total weight' and 'Weight' show one bar weight. Use the NUMBER_OF_BARS_IN_GROUP and WEIGHT_TOTAL_IN_GROUP properties to export the values of the group to the IFC file.

This advanced option is model specific and the setting is saved in the options database.

XS_EXPORT_LINE_TYPE_DEFINITION_FILE

Category

Export

Enter the name of the line type definition file that contains the line type definitions and is used in line type mapping.

The file name extension of the line type definition file is `.lin`. The advanced option is set to point to the file `TeklaStructures.lin` by default.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_EXPORT_STEEL2000_PRIMARY_IDS

Category

Drawing Properties

Set to `TRUE` to have Tekla Structures include main part IDs in MIS export files. IDs appear on separate rows in the file. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_EXTENSION_DIRECTORY

This advanced option must be set in an initialization (.ini) file.

Use the advanced option `XS_EXTENSION_DIRECTORY` for defining additional installation folders for extensions, or customer tools developed on top of Open API.

Before the additional installation folders, the default folder `%XSDATADIR%\environments\common\extensions` is used in installation.

XS_EXTERNAL_EXCEL_DESIGN_PATH

This advanced option must be set in an initialization (.ini) file.

This advanced option is system specific and is read from `teklastructures.ini`. It can also be set locally, see your environment `.ini`

file (`env_<environment_name>.ini`). Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

Points to the location of the Excel spreadsheet used in connection design. If you want to change the location, set the advanced option in the `user.ini` file.

1.6 Advanced options - F

XS_FILTER_SEPARATOR_CHAR

Category

Modeling properties

Enter the separator to be used between filter strings, for example, in the view filter. You can use any character. By default Tekla Structures uses a space.

Example

To use a semi-colon as a separator, set this advanced option as follows:

```
XS_FILTER_SEPARATOR_CHAR=;
```

This advanced option is model specific and the setting is saved in the options database.

XS_FIRM

Category

File Locations

NOTE This advanced option is system-specific and only meant for administrators.

Set the advanced options `XS_PROJECT` and `XS_FIRM` along with `XS_SYSTEM` to point to the folders Tekla Structures searches for property files. Tekla Structures always saves properties in the current `model\attributes` folder. You can then copy or move them to the `XS_FIRM` or the `XS_PROJECT` folder if the same settings are needed in other models. You can also create user-defined sub-folders under the `XS_FIRM` and the `XS_PROJECT` folders, and copy or move property files from the `model\attributes` folder to these sub-folders.

For shared models, you can use a sub-folder in a Trimble Connect project as the firm folder. For more information, see .

Note that you can define one path only, not a list of paths.

WARNING Changing an advanced option value in `.ini` files located outside the model folder does not affect the existing models. You can only update advanced options in the **Advanced Options** dialog box or in the `options.ini` file located in model folder; not from an `options.ini` file located in folders defined for the advanced options `XS_FIRM` or `XS_PROJECT`. The `.ini` files are read also when you open an existing model, but only new advanced options that do not exist in `options_model.db` or `options_drawings.db` are inserted, for example, such options that are not yet in the **Advanced Options** dialog box but have been added in the software.

See also

XS_FIX_FRAME_OF_FIXED_MODELVIEW

Category

Drawing Properties

This advanced option prevents the changing of the view size and location especially in general arrangement drawings.

Set to `TRUE` to fix the frames of views that have **Placing** set to **Fixed** in **View Properties**. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_FLAT_PREFIX

Category

Plate Work

Use this advanced option to set the flat bar prefix for output, reports, and marks. If Tekla Structures finds a matching flat bar in the `fltprops.inp` file, the plate name will consist of the prefix you enter here, followed by the thickness X width, for example FLAT5X100. The default value is `FLAT`.

By default PL and PLT profiles get the prefix FL or FLT if a matching plate is found in the `fltprops.inp`.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is

in use, you can change the value, which is then the same for all users in the current model.

NOTE In Imperial version, if `XS_FLAT_PREFIX` is not included in `profitab.inp` as similar profile as PL (valid parametric profile name), the profile is shown with metric units.

XS_FLAT_THICKNESS_TOLERANCE

Category

Plate Work

Use to set the checking tolerance for flat bars. Tekla Structures uses this value to check plate thickness to determine whether to convert it to a flat bar. Enter a decimal value. The default value is 0.1 mm.

This advanced option is model specific and the setting is saved in the options database.

XS_FLAT_TOLERANCE

Category

Plate Work

Tekla Structures uses this value to check plate width to determine whether to convert it to a flat bar. Enter a decimal value. The default value is 0.1 mm.

This advanced option is model specific and the setting is saved in the options database.

FLEXLM_TIMEOUT

Category

This advanced option must be set in an initialization (.ini) file.

This is a Windows environment variable used by Tekla Structures. This variable reduces the delay in starting Tekla Structures. Enter the value in microseconds. For Tekla Structures, the maximum value of this variable is `100 000`.

Example

```
set FLEXLM_TIMEOUT=100000
```

XS_FRACTION_HEIGHT_FACTOR

Category

Imperial Units

Use this advanced option to control the total height of fractions. The default value in the US environment Imperial role is `1.3`.

This advanced option is model specific and the setting is saved in the options database.

XS_FS_POSTFIX_FOR_MERGED_PART_MARK

Category

Marking: Parts

Use to define the far side postfix in merged part marks. This postfix is visible for identical parts on far side. FS is the default value.

This advanced option is model specific and the setting is saved in the options database.

See also

1.7 Advanced options - G

XS_GA_CONNECTING_SIDE_MARK_SYMBOL

This advanced option must be set in an initialization (.ini) file.

Use to change the symbol for a connecting side mark in general arrangement drawings. By default the side mark symbol is number 34 in the symbol file. To change the symbol, set this advanced option to a different symbol number.

Restart Tekla Structures after changing the value to activate the new setting.

See also

XS_CONNECTING_SIDE_MARK_SYMBOL (page 129)

XS_GA_DRAWING_VIEW_TITLE

Category

Drawing Properties

Use to define a title for a general arrangement drawing view in a multidrawing. By default the value is defined as follows:

`Drawing %DRAWING_BASE_NAME%`.

This advanced option is model specific and the setting is saved in the options database.

XS_GAGE_OF_OUTSTANDING_LEG_STRING

Category

Marking: Bolts

Use to display the gage of outstanding leg (GOL) information in bolt marks (element **Gage of outstanding leg**) in the following ways:

- If there is no `%VALUE%` in the string then Tekla Structures adds the GOL value to the end of the string.
- You can enter any combination of text together with option `%VALUE%`. For example, `%VALUE% GOL` or `GOL%VALUE%`.
- If you do not set this advanced option, Tekla Structures only uses `%VALUE%`.
- If Tekla Structures cannot calculate the value, nothing is added to the mark.
- The default value is `GOL%VALUE%`.

This advanced option is model specific and the setting is saved in the options database.

Example

Here, the value for the gage of outstanding leg is 5½.

In the Advanced Options dialog box	In bolt mark
%VALUE% GOL	5½ GOL
GOSL %VALUE%	GOSL 5½
GOL =	GOL = 5½
	5½

XS_GA_HIDDEN_NORTH_MARK_SYMBOL

Category

Marking: Parts

By default, the symbol for hidden North marks in general arrangement drawings is number 32 in the symbol file. To change the symbol, set this advanced option to a different symbol number.

See also

Show orientation marks (north marks)

XS_GA_NORTH_MARK_SCALE

Category

Marking: Parts

By default, Tekla Structures draws North marks in general arrangement drawings using a scale of 1:1. Use this advanced option to specify a different scale. You can also create a larger symbol for North marks in the Symbol Editor.

This advanced option is model specific and the setting is saved in the options database.

XS_GA_NORTH_MARK_SYMBOL

Category

Marking: parts

By default, the symbol for North marks in general arrangement drawings is number 32 in the symbol file. To change the symbol, set this advanced option to a different symbol number.

This advanced option is model specific and the setting is saved in the options database.

See also

Show orientation marks (north marks)

XS_GA_OMITTED_DIAMETER_TYPE

Category

Marking: bolts

Use to omit marks for specific diameter types in general arrangement drawings. The options are `HOLE` or `BOLT`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_OMITTED_BOLT_TYPE (page 334)

XS_GET_ASSEMBLY_LEVELS_FROM_ASSEMBLY_MAIN_PART

Category

Marking: Parts

Set this advanced option to `TRUE` to get assembly levels from the assembly main part. Set it to `FALSE` to get the levels from the whole assembly. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_GET_CAST_UNIT_LEVELS_FROM_CAST_UNIT_MAIN_PART

Category

Marking: Parts

Set this advanced option to `TRUE` to get cast unit levels from the cast unit main part. Set it to `FALSE` to get the levels from the whole cast unit. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_GOL_SYMMETRY_DISTANCE

Marking: Bolts

Gauge of outstanding leg (GOL) is the center to center distance of the holes between two angles usually connected on the web of the beam/column. It is

equal to gauge of leg angles plus the web thickness. Outstanding leg is the leg of the angle perpendicular to the paper viewing from the web. Use this advanced option to set the tolerance in part symmetry checking when calculating the gauge of outstanding leg. The default value is 0.01.

This advanced option is model specific and the setting is saved in the options database.

XS_GRID_DIMENSION_OVERALL_LENGTH

Category

Dimensioning: General

Set a length to add a dimension line spanning the entire grid next to existing grid dimensions. The default value is `1`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_GRID_COLOR_FOR_WORK_PLANE

Category: Model View

Use to change the color of the work plane grid in the model. Define the color of the grid using RGB values:

`<value for red> <value for green> <value for blue>`.

Separate the values with spaces. Define the values on a scale of 0 to 1. The default values are `0.7 0.0 0.3`.

Reopen the model view to activate the new value.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`.

XS_GRID_PLANES_VISIBLE_WITH_USERPLANES

Category

Model View

Use to show or hide the grid planes.

Set to `TRUE` to show the grid planes. Set to `FALSE` (default) to hide the grid planes.

Reopen the view for the change to take effect.

NOTE The grid planes can be shown only if the construction planes are visible. To display the construction planes, select the **Construction planes** check box in the **Display** dialog box.

XS_GRID_TEXT_FONT

Category

Drawing Properties

Use to set the font for grid text. The default values is Arial. If you do not specify a font, Tekla Structures uses the default font defined for `XS_DEFAULT_FONT`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DEFAULT_FONT (page 144)

1.8 Advanced options - H

XS_HANDLE_SCALE

Category: Model view

Use to modify the handle size in model views. Enter a decimal value.

- The default value is `1.3`.
- A value larger than 1.3 makes the handles larger and easier to see.
- A value smaller than 1.3 decreases the handles.

Close the model view and re-open it to implement the change.

This advanced option is user specific, and the setting is saved in `options.bin` under the user folder.

XS_HATCH_PATTERN_LINE_LIMIT

Category: **Hatching**

Set this advanced option to a numerical value to control the maximum number of line segments created by a hatch pattern. The default value is 0, which means that there is no limit.

This advanced option is model specific and the setting is saved in the options database.

XS_HATCH_SCALE_LIMIT

Category

Hatching

Defines the smallest possible size of the drawn hatch pattern. If the scale of a single hatch pattern is smaller than the defined value, the pattern is changed to a solid face. The default value is `0.001`.

This advanced option is model specific and the setting is saved in the options database.

XS_HATCH_SEGMENT_BUFFER_SIZE

Category

Hatching

Tekla Structures includes a hatch buffer to speed up opening drawings containing hatch. This advanced option defines the size of the buffer.

The default value is 1000000. If you use extremely complicated hatches, you may get better performance with a larger value. For small hatches, use a smaller value.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_HATCH_SPECIAL_COLOR_ACI

Category

Export

Use this advanced option to set the autocad color index for the **Special** color in hatches in drawing export. The default values is `120`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_HATCH_SPECIAL_COLOR_R

Category

Hatching

Use to define a hatch color that is not converted to black when printed. This color will be printed as color or grayscale, depending on the selected printer settings. The hatch color is defined using RGB (Red Green Blue) values on a scale of 0 to 255. The default for all of the related advanced options is 230.

Define the color using the following advanced options:

`XS_HATCH_SPECIAL_COLOR_R`

`XS_HATCH_SPECIAL_COLOR_G`

`XS_HATCH_SPECIAL_COLOR_B`

The smaller the values, the darker the shade.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_HATCH_SPECIAL_COLOR_G

Category

Hatching

See XS_HATCH_SPECIAL_COLOR_R (page 270)

XS_HATCH_SPECIAL_COLOR_B

Category

Hatching

See XS_HATCH_SPECIAL_COLOR_R (page 270)

XS_HELP_PATH

Category

This advanced option is available only in the `lang_<CurrentLanguage>.ini` **files.**

This advanced option specifies the location of the help files in `chm` (Microsoft Compiled HTML Help) format. These help files are used for some components.

This advanced option is system specific and is read from environment files. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

Example

```
set XS_HELP_PATH=%XSDATADIR%\help\enu
```

XS_HIDDEN_LINES_CHECK_TOLERANCE

Category

Drawing Properties

Defines the distance within which Tekla Structures treats part lines inside cast units as overlapping lines. The default value is `0.01`.

This advanced option is model specific and the setting is saved in the options database.

XS_HIDDEN_LINES_UNHIDE_EMBEDDED

Category: Drawing Properties

Set this advanced option to `TRUE` to draw the edge lines of embedded parts as visible lines even when they are hidden inside concrete. `FALSE` is the default value.

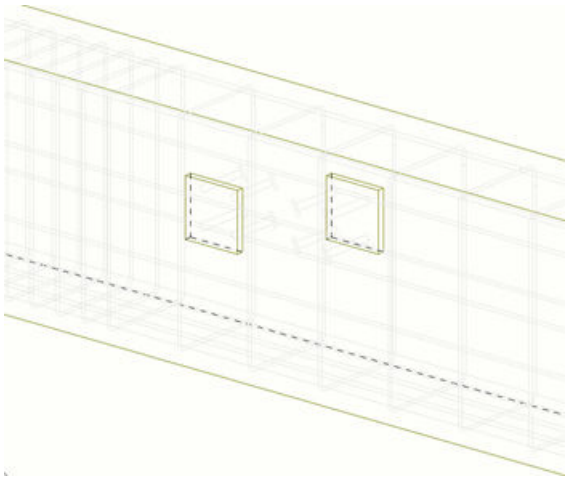
Embed is a non-concrete (steel or miscellaneous) object in a cast unit or pour. It can be a sub-assembly, like steel objects always are in cast units, or a part in a pour.

Note that this advanced option does not affect reinforcement.

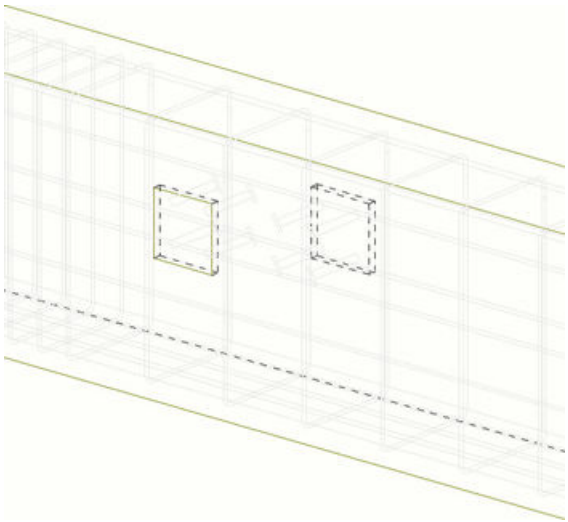
In the following two examples, one of the embeds is in the front/near face and the other is in the back/far face of the beam.

In the first example:

- XS_HIDDEN_LINES_UNHIDE_EMBEDDED is set to TRUE, so embed part lines are visible also when they are inside concrete.
- Cast unit lines are shown as outlines.
- Embed anchor studs have been created as rebars, and they are hidden, because **Hide lines behind parts** has been selected in reinforcement properties.



In the second example, XS_HIDDEN_LINES_UNHIDE_EMBEDDED is set to FALSE, so embed part lines that are inside concrete are hidden.



This advanced option is model specific and the setting is saved in the options database.

XS_HIDDEN_NORTH_MARK_SYMBOL

Category

Marking: parts

By default hidden North mark symbol is number 32 in the symbol file. To change the symbol, set this advanced option to a different symbol number.

This advanced option is model specific and the setting is saved in the options database.

See also

Show orientation marks (north marks)

XS_HIDDEN_REMOVE_DOUBLE_LINES

Category

Drawing Properties

To have Tekla Structures draw double lines when displaying models with **Exact** representation and creating drawings and 2D DXF files, set this advanced option to `FALSE`.

The default value is `TRUE`, which means that Tekla Structures does not draw double lines, to minimize file size.

This advanced option is model specific and the setting is saved in the options database.

XS_HIDDEN_USE_BOLT_PLANES

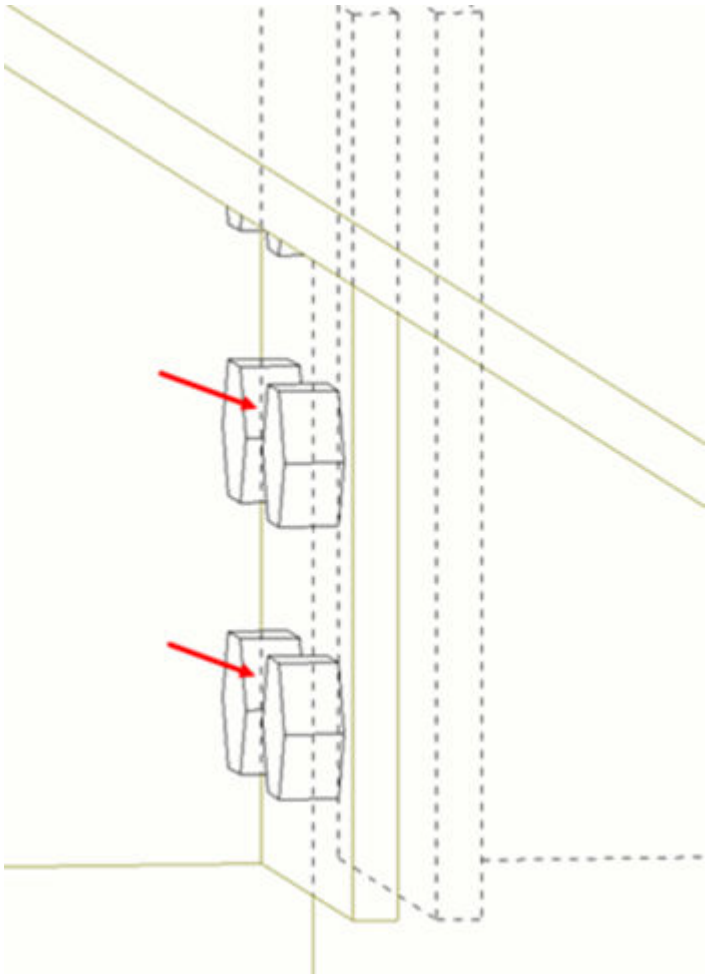
Category

CategoryDrawing Properties

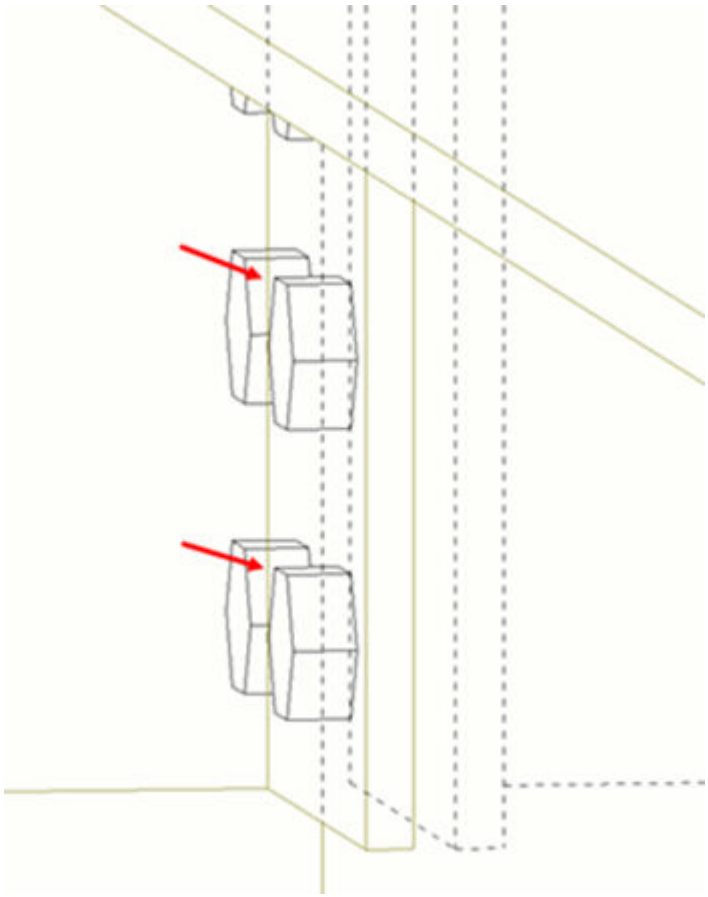
Set this advanced option to `TRUE` to hide lines behind a bolt nut (with **Exact** part representation). To show the lines, set it to `FALSE` (default).

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

`TRUE`:



FALSE:



XS_HIDE_OTHER_PARTS_IN_ASSEMBLY_AND_CAST_UNIT_VIEWS

Category

Model View

Defines whether parts that do not belong to assemblies and cast units are shown or hidden in assembly and cast unit views. The default value `TRUE` hides the parts that do not belong to the selected assembly or cast unit. If you set this advanced option to `FALSE`, the parts are not hidden.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

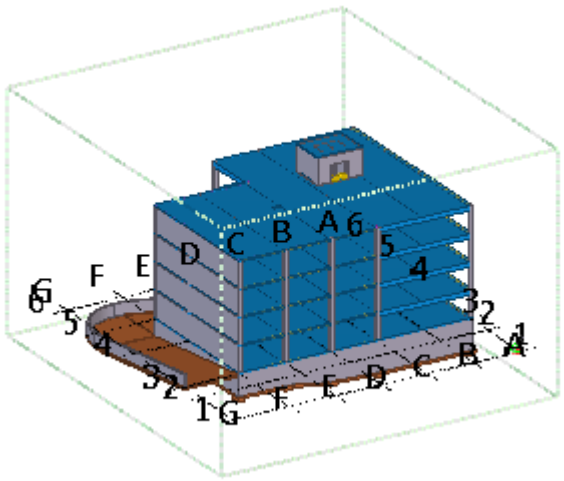
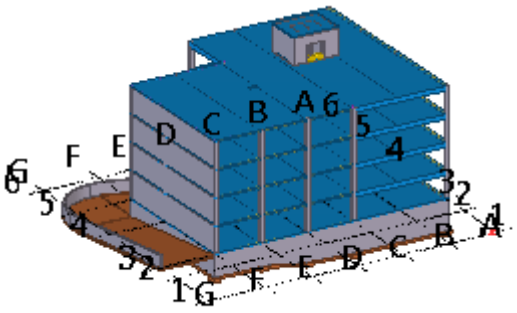
XS_HIDE_WORKAREA

Category: Model View

Set this advanced option to `TRUE` to hide the work area or to `FALSE` to display the work area in model views. The default is `FALSE`. Redraw the views for the change to take effect.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

Example

Setting	Appearance in model
`FALSE`	
`TRUE`	

TIP To temporarily hide the green work area box, hold down the **Ctrl** and **Shift** keys simultaneously, right-click and select **Redraw View**. To make the box visible again, right-click and select **Redraw View** again.

XS_HIGHLIGHT_ASSOCIATIVE_DIMENSION_CHANGES

Category

Dimensioning: General

Use to define whether changed dimension text and moved dimension points are highlighted in associative drawings.

When set to `TRUE` (default), Tekla Structures highlights the changed dimension text and the moved dimension points in updated associative drawings.

Tekla Structures highlights the changes in the following ways:

- A change symbol (by default a cloud) is drawn around the old point, the new point and the dimension values. It is displayed only when you select the dimension.
- An arrow is drawn from the old point to the new point.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_HIGHLIGHT_MARK_CONTENT_CHANGES (page 277)

XS_ASSOCIATIVE_CHANGE_HIGHLIGHT_SYMBOL (page 89)

XS_ASSOCIATIVE_CHANGE_HIGHLIGHT_SIZE (page 89)

XS_HIGHLIGHT_MARK_CONTENT_CHANGES

Category

Marking: General

Use to define whether changed mark contents are highlighted in associative drawings.

When set to `TRUE` (default), Tekla Structures highlights the mark contents that have been changed in updated associative drawings by drawing a change symbol (by default a cloud) around the changed mark content.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_HIGHLIGHT_ASSOCIATIVE_DIMENSION_CHANGES (page 277)

XS_ASSOCIATIVE_CHANGE_HIGHLIGHT_SYMBOL (page 89)

XS_ASSOCIATIVE_CHANGE_HIGHLIGHT_SIZE (page 89)

XS_HOLE_MARK_STRING_FOR_SIZE

Category

Marking: Bolts

Use to define the contents of the **Size** element in hole marks. The default value is `%BOLT_NUMBER%*D%HOLE.DIAMETER%`.

This advanced option is only used when there is a hole, no bolt (and the hole is a normal one).

The advanced options `XS_SHOP_HOLE_MARK_STRING_FOR_SIZE` and `XS_SITE_HOLE_MARK_STRING_FOR_SIZE` override this setting.

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)
- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD
- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK
- BOLT_SHORT_NAME

- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SITE_HOLE_MARK_STRING_FOR_SIZE (page 433)

XS_SHOP_HOLE_MARK_STRING_FOR_SIZE (page 411)

XS_HOLE_MARK_STRING_FOR_SIZE_IN_GA

Category

Marking: Bolts

Use to define the contents of the size element in bolt marks in general arrangement drawings. If you have not set the advanced options `XS_SHOP_HOLE_MARK_STRING_FOR_SIZE_IN_GA` or `XS_SITE_HOLE_MARK_STRING_FOR_SIZE_IN_GA`, then this advanced option is used.

This advanced option is only used when there is a hole, no bolt (and the hole is a normal one).

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)
- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD
- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE

- BOLT_COUNTERSUNK
- BOLT_SHORT_NAME
- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SHOP_HOLE_MARK_STRING_FOR_SIZE_IN_GA (page 412)

XS_SITE_HOLE_MARK_STRING_FOR_SIZE_IN_GA (page 434)

1.9 Advanced options - I

XS_IGNORE_CUT_VALUE_IN_TEMPLATE

This advanced option must be set in an initialization (.ini) file.

By default, when a Template Editor template output value field does not fit into a cell, three asterisks (***) will indicate that the value is cut. For example, when a template has a value field length limited to 10 characters and the output value is 11 characters or more, *** will be displayed at the end of the value.

If you do not want to display the asterisks, set `XS_IGNORE_CUT_VALUE_IN_TEMPLATE` to `TRUE`.

XS_IGNORE_CROSSBAR_LOCATION_IN_REBAR_MESH_NUMBERING

Category: Numbering

Use this advanced option to define whether the location of the crossing bars (e.g. above or below the main bars) is ignored in the numbering of reinforcement meshes.

The default value is `FALSE`.

If `XS_IGNORE_CROSSBAR_LOCATION_IN_REBAR_MESH_NUMBERING` is set to `FALSE`, the location of the crossing bars affects numbering, and the otherwise identical meshes receive different numbers if the crossing bars are on the different sides of the main bars.

If XS_IGNORE_CROSSBAR_LOCATION_IN_REBAR_MESH_NUMBERING is set to TRUE, crossing bars can be on either side of the main bars, and the otherwise identical meshes still receive the same number.

This advanced option is model specific and the setting is saved in the options database.

NOTE We do not recommend that you change this setting during a project. If you do so, you must carry out full numbering (**File** --> **Diagnose & repair** --> **Diagnose and repair numbering: all**).

XS_IGNORE_SUBASSEMBLY_HIERARCHY_IN_DIMENSIONING

Category

Dimensioning: Parts

Use to define whether sub-assembly parts are dimensioned as secondary parts of the main assembly.

When set to TRUE, Tekla Structures ignores sub-assemblies, and parts inside the sub-assemblies are dimensioned as if they were parts in the main assembly. The default value is FALSE.

This advanced option is model specific and the setting is saved in the options database.

XS_IFC_EXPORT_OBJECT_LAYER_FROM_UDA

This advanced option must be set in an initialization (.ini) file.

Use this advanced option to define for the IFC2x3 export the layer name of an object in an IFC file using a predefined UDA. Using this advanced option requires that you have selected the option **Layer names as part names** in the IFC2x3 export settings. By default, the value is empty, and the part name is used as the layer name.

The UDA name is case sensitive, so you need to check how the UDA name is written in the objects.inp file. By default, the objects.inp file is located in C:\ProgramData\Tekla Structures\Work\Environments\common\inp.

Enter the following in an .ini file:

```
set XS_IFC_EXPORT_OBJECT_LAYER_FROM_UDA=<UDA>
```

Replace the <UDA> with the UDA of your choice.

Example: The UDA input field “Fabricator name” is defined as “fabricator” in `objects.inp`, so enter the UDA as follows:

```
set XS_IFC_EXPORT_OBJECT_LAYER_FROM_UDA=fabricator
```

XS_IFC2X3_EXPORT_SECONDARY_AS_DISCRETEACCESSORY

This advanced option must be set in an initialization (.ini) file.

The IFC2x3 export does not export steel secondary objects as IfcDiscreteAccessory. You can force steel secondary objects to the IfcDiscreteAccessory category by setting this advanced option to `TRUE` in an `.ini` file, for example, in `teklastructures.ini`.

XS_IMPERIAL

Category

Imperial Units

Set this advanced option to `TRUE` to have output in imperial units only. If you do not want to do this, set it to `FALSE` (default).

This affects only the following elements of bolt marks:

- Gage of outstanding leg (GOL)
- Center-to-center distance

This advanced option is model specific and the setting is saved in the options database.

XS_IMPERIAL_DATE

Category

Imperial units

Set this advanced option to `TRUE` to use the date format mm/dd/yyyy. Set it to `FALSE` (default) to use the date format dd.mm.yyyy.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_IMPERIAL_INPUT

Category

Imperial Units

Set this advanced option to `TRUE` to allow input in imperial units only. To disable the advanced option, set it to `FALSE` (default).

This advanced option is model specific and the setting is saved in the options database.

XS_IMPERIAL_TIME

Category

Imperial Units

Set this advanced option to `TRUE` to use the time format `hh:mm:ss am/pm`.

Set to `FALSE` to use the time format `hh:mm:ss`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_IMPERIAL_TRIANGLES

Category

Imperial Units

Set this advanced option to `TRUE` to also show triangle ratios in inches.

The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_IMPORT_DWG_TEXT_AS_POLYGON

Category: Drawing Properties

This advanced option can be used when adding DWG files to drawings. If you have problems with fonts in DWG files, set this advanced option to `TRUE`, and

Tekla Structures will import fonts as polygons, instead of fonts. By default, the option is set to `FALSE`.

This advanced option is user specific, and the setting is saved in `options.bin` under the user folder.

XS_IMPORT_MODEL_LOG

Category

Import

Set to `TRUE` to create a log every time you use an import model.

Set to `APPEND` to append a log entry to the previous log.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_INCH_SIGN_ALWAYS

Category

Imperial Units

By default, Tekla Structures does not put an inch symbol (") in dimensions which contain only inches. Set this advanced option to `TRUE` to show inch symbols in all dimensions. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_INCLUDE_DWG_ATTRIBUTES_IN_REPORTS_AND_INQUIRE

Category

Speed and Accuracy

DWG-specific attributes are now available in reports and inquiries only if this advanced option is set to `TRUE`. This advanced option is by default set to `FALSE`.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_INHERIT_CONCRETE_PART_NUMBERING_SETTINGS_FROM_CAST_UNIT

Category: Numbering

Use this advanced option to define whether to set concrete part numbering according to cast unit numbering settings. If you set this advanced option to `TRUE` (default), the part number prefix for concrete parts includes the cast unit prefix and the start number. If you set the advanced option to `FALSE`, the cast unit prefix and the start number are not included.

For example, the cast unit prefix is `C` and the start number is `100`. When the advanced option is `TRUE`, the concrete part prefix is `Concrete_C-100`. When the advanced option is `FALSE`, the prefix is only `Concrete`.

This setting affects concrete parts: Strip and pad footings, concrete beams and columns, concrete walls and slabs, and concrete polybeams.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_CONCRETE_PART_NUMBERING_PREFIX (page 129)

XS_CONCRETE_PART_NUMBERING_START_NUMBER (page 129)

XS_INP

This advanced option is system specific and is read from `teklastructures.ini`. It can also be set locally, see your environment `.ini` file (`env_<environment_name>.ini`). Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

Several system files are read from this folder, such as definitions for parametric profiles (`.clb`), IFC property set configurations (`.xml`), line type mappings (`.lin`), global UDA definitions (`objects.inp`), and access rights (`privileges.inp`).

The default location is `XSDATADIR%\environments\common\inp\`. You can use semicolon-separated lists of folder paths.

XS_INTELLIGENCE_DO_NOT_REMOVE_OBSOLETE_VIEWS

Category

Drawing Properties

Enter `TRUE` in the **Value** field to prevent drawing views from being deleted when associated objects are removed from the model. `FALSE` is the default.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

For general arrangement drawings use the advanced option `XS_INTELLIGENCE_DO_NOT_REMOVE_OBSOLETE_VIEWS_IN_GA`.

See also

XS_INTELLIGENCE_DO_NOT_REMOVE_OBSOLETE_VIEWS_IN_GA (page 286)

XS_INTELLIGENCE_DO_NOT_REMOVE_OBSOLETE_VIEWS_IN_GA

Category

Drawing Properties

Enter `TRUE` (default) to prevent drawing views from being deleted when associated objects are removed from the model.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

See also

XS_INTELLIGENCE_DO_NOT_REMOVE_OBSOLETE_VIEWS (page 285)

XS_INTELLIGENCE_MAX_PART_COUNT

Category: **Drawing Properties**

Enter an integer to define how many parts are taken into account when searching for associated objects. The advanced option is set to `20` by default.

This advanced option is model specific and the setting is saved in the options database.

XS_INTELLIGENCE_MAX_PLANE_COUNT

Category:Drawing Properties

Enter an integer to define how many planes are taken into account when searching for associated objects. The advanced option is set to `1000` by default.

This advanced option is model specific and the setting is saved in the options database.

XS_INTELLIGENCE_MAX_RULE_COUNT

Category: **Drawing Properties**

Use to decrease the number of associative rules that are used for one measurement point. A smaller value, for example `10` (default), is usually enough. A smaller value may also increase performance and decrease the database size.

This advanced option also controls the maximum number of shown rules in the dimension associativity rule list.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

WARNING Use this advanced option only when there are performance issues with associative drawings. Using this advanced option may cause losing of associativity in some cases when objects have been deleted from the model.

XS_INTELLIGENT_CLONING_ADD_DIMENSIONS

Category

Drawing Properties

Set to `TRUE` to create dimensions for additional parts using automatic dimensioning during cloning, and to `FALSE` to prevent this. The default is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

XS_INTELLIGENT_DRAWING_ALLOWED

Category

Drawing Properties

Clear the value or set it to `FALSE` to prevent Tekla Structures from automatically moving dimensions, marks, etc. according to model changes. `TRUE` is the default value.

This advanced option is model specific and the setting is saved in the options database.

To control how the drawing view placing works when the model changes, set this advanced option to `TRUE` and use it together with the advanced option . `XS_DRAWING_UPDATE_VIEW_PLACING`.

NOTE This setting affects all drawings. To prevent Tekla Structures from automatically updating general arrangement drawings, use the advanced option `XS_INTELLIGENT_DRAWING_ALLOWED_IN_GA`.

See also

XS_INTELLIGENT_DRAWING_ALLOWED_IN_GA (page 288)

XS_DRAWING_UPDATE_VIEW_PLACING (page 224)

XS_INTELLIGENT_DRAWING_ALLOWED_IN_GA

Category

Drawing Properties

Clear the value or set the advanced option to `FALSE` to prevent Tekla Structures from automatically moving dimensions, marks, etc. according to model changes.

This advanced option is model specific and the setting is saved in the options database.

NOTE This setting affects only general arrangement drawings. To prevent Tekla Structures from automatically updating all types of drawings, use the advanced option `XS_INTELLIGENT_DRAWING_ALLOWED`.

See also

XS_INTELLIGENT_DRAWING_ALLOWED (page 287)

XS_INTELLIGENT_MESSAGES_ALLOWED

Category

Drawing Properties

Set this advanced option to `TRUE` to display message when a drawing is opened if a model object to which one or more drawing objects are linked is deleted from the model.

If you do not want to display the message, set this advanced option to `FALSE` (default).

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_INTELLIGENT_UPDATE_ADD_DIMENSIONS

Category

Drawing Properties

Set to `TRUE` to add dimensions for new parts, bolts and reinforcing bars when updating drawings and to `FALSE` to prevent this. The default is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

XS_INVALID_POUR_BREAK_COLOR

This advanced option must be set in an initialization (.ini) file.

Use this advanced option to change the color of invalid pour breaks in model views. Enter number as the value using the class numbers in the part properties in the property pane to indicate the color. For example, if you set this advanced option to 6, Tekla Structures will color all invalid pour breaks yellow. The default value is 58 (red).

See also

IS_POUR_BREAK_VALID (page 562)

XS_I_PROFILE_CENTER

Category

Dimensioning: Parts

Set to `NONE` to prevent Tekla Structures from using the center line of I profiles to dimension front views. By default, no value is set.

This advanced option is model specific and the setting is saved in the options database.

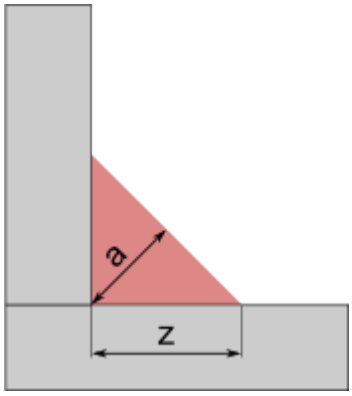
See also

XS_ISO_LEG_LENGTH_AS_WELDSIZE

Category: Welds

When XS_AISC_WELD_MARK (page 72) is set to `FALSE`, set `XS_ISO_LEG_LENGTH_AS_WELDSIZE` to:

- `TRUE` to use the weld size as the leg length (z) of the fillet welds.
- `FALSE` to use the weld size as the throat thickness (a) of the fillet welds.



NOTE The weld prefix overrides the setting of the advanced option `XS_ISO_LEG_LENGTH_AS_WELDSIZE`. If you have set the prefix of a weld to `a`, the weld size determines the design throat thickness. When the weld prefix is `z`, the weld size determines the leg length.

If the last character of the weld prefix is `s`, Tekla Structures creates the solid weld object so that `a` equals weld size.

This advanced option is model specific and the setting is saved in the options database.

After changing this setting, modify existing welds or their properties to activate the new setting and to update the weld solids.

1.10 Advanced options - J

XS_JOINT_NUMBER_FORMAT

Category

Marking: General

The connection numbers are visible in drawings, when you set the **Connection mark** field to **Number** in the **Connection mark properties** dialog box. Use this advanced option to define the connection number format. For example, you can use it to define prefix text.

This advanced option is model specific and the setting is saved in the options database.

Example

In `XS_JOINT_NUMBER_FORMAT=J%3.3d`:

- `J` is the prefix.

  The rest of the string defines the number format.
- The first number defines the minimum field width.
- The second number defines the minimum quantity of numbers to display.
- `%` and `d` (integer value) indicate the format.

XS_JOINTS_USE_NOTCH1

Category

Components

Set to `1` to have connections use standard notch routines. This is the default value.

Set to `0` to have connections use simple notch routines.

This advanced option is model specific and the setting is saved in the options database.

1.11 Advanced options - K

XS_KEEP_AUTOSAVE_FILES_ON_EXIT_WHEN_NOT_SAVING

Category

Modeling Properties

Tekla Structures deletes autosave files when you close a model to save disk space. If this advanced option is set to `TRUE`, Tekla Structures does not delete these files, even if you exit Tekla Structures without saving the model. The default value is `TRUE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_KEYIN_ABSOLUTE_PREFIX

Category: Modeling Properties

Defines the character used in absolute snapping. Enter any allowed ASCII character. The default value is $.

If you have set Tekla Structures to use absolute snapping by default using the advanced option `XS_KEYIN_DEFAULT_MODE`, you do not need to use a snap character for absolute snapping.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_KEYIN_DEFAULT_MODE (page 292)

XS_KEYIN_RELATIVE_PREFIX (page 293)

XS_KEYIN_GLOBAL_PREFIX (page 293)

XS_KEYIN_DEFAULT_MODE

Category: Modeling Properties

Defines the snapping mode Tekla Structures uses by default. Set to `RELATIVE` (default), `ABSOLUTE`, or `GLOBAL`.

In the relative snapping mode, the coordinates you enter in the **Enter a Numeric Location** dialog box as such without any prefix are relative to the last position picked.

In the absolute snapping mode, the coordinates are based on the origin of the work plane.

In the global snapping mode, the coordinates are based on the global origin and the global x and y directions.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_KEYIN_RELATIVE_PREFIX (page 293)

XS_KEYIN_ABSOLUTE_PREFIX (page 292)

XS_KEYIN_GLOBAL_PREFIX (page 293)

XS_KEYIN_GLOBAL_PREFIX

Category: Modeling Properties

Defines the character used in global snapping. The default value is `!`.

If you have set Tekla Structures to use global snapping by default using the advanced option `XS_KEYIN_DEFAULT_MODE`, you do not need to use a snap character for global snapping.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_KEYIN_DEFAULT_MODE (page 292)

XS_KEYIN_RELATIVE_PREFIX (page 293)

XS_KEYIN_ABSOLUTE_PREFIX (page 292)

XS_KEYIN_RELATIVE_PREFIX

Category: Modeling Properties

Defines the character used in relative snapping. Enter any allowed ASCII character. The default value is `@`.

If you have set Tekla Structures to use relative snapping by default using the advanced option `XS_KEYIN_DEFAULT_MODE`, you do not need to use a snap character for relative snapping.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_KEYIN_DEFAULT_MODE (page 292)

XS_KEYIN_ABSOLUTE_PREFIX (page 292)

XS_KEYIN_GLOBAL_PREFIX (page 293)

XS_KNOCK_OFF_DIMENSION_PRECISION

Category

Dimensioning: General

Set this advanced option to 16 or 32 to set the precision of the knock-off dimensions to 1/16 or 1/32. Otherwise the precision is the one that is set in the drawing level dimensioning dialog box. The default value is zero. Other values are ignored, and knock off dimension precision is the same as for other dimensions.

1.12 Advanced options - L

XS_LANGUAGE

This advanced option must be set in an initialization (.ini) file.

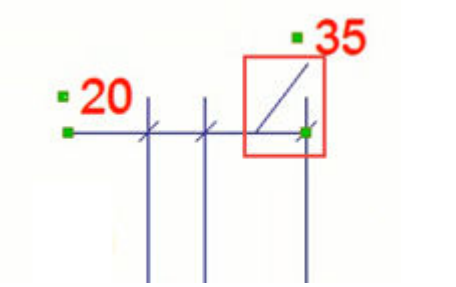
This advanced option is system specific and is read from environment files. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

This advanced option sets the Tekla Structures default language. The default language is displayed first when you go to the **File** menu and click **Settings** --> **Change language** .

XS_LEADER_LINE_TO_DRAGGED_DIMENSION_TEXT

Category: Dimensioning: General

When you set this advanced option to `TRUE`, a leader line is drawn when a dimension text is dragged away from the dimension line. When you set it to `FALSE`, the leader line is not drawn. The default value is `TRUE`.



This advanced option is model specific and the setting is saved in the options database.

XS_LICENSE_SERVER_HOST

This advanced option must be set in an initialization (.ini) file.

With this advanced option the administrator can predefine the license server address for the user, so that the user does not need to make the selection and type the on-premises license server port and host name in the licensing dialog box at the first start of the Tekla Structures software.

You can add this advanced option in a customized `.ini` file and use it in start-up shortcuts for starting `teklastructures.exe` with the initialization that you have customized, for example.

For online licensing, enter `https` as the value.

Example

```
set XS_LICENSE_SERVER_HOST=https
```

For on-premises licensing, enter the advanced option value in one of the following formats:

```
port@host
```

```
port@ip-address
```

Example

```
set XS_LICENSE_SERVER_HOST=27007@MY_LICENSE_SERVER_NAME
```

XS_LOAD_MODELING_CODE

This advanced option must be set in an initialization (.ini) file. For more information about initialization files, see .

Use this advanced option for defining the set of load group types. Load combinations are generated according to rules that are specific to the load modeling code. The default value is `EuroCode`.

Possible values are: `EuroCode`, `AISC`, `UBC`, `IBC`, `ACI`, `BS`, `CM66 (F)`, and `BAEL91 (F)`.

Example

```
set XS_LOAD_MODELING_CODE=EuroCode
```

XS_LOG_FILE_NAME

Category

This advanced option must be set in an initialization (.ini) file.

This advanced option is system specific and is read from environment files. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

This advanced option defines the name of the Tekla Structures log file. The default value is `TeklaStructures.log`.

See also

XS_LOG_LEVEL

Category: Speed and Accuracy

Use this advanced option to define which messages Tekla Structures writes to the session history log file, `TeklaStructures_<user>.log`. Set the level of the log output to one of the following options:

- `DEBUG`: all log messages are written
- `INFO`: all log messages except debug messages are written
- `WARNING`: all log messages except debug and info messages are written
- `ERROR`: only error and assert messages are written

The default value is `INFO`. Providing any value other than one of the above results in log level `INFO`.

This advanced option is system specific and is read from `teklastructures.ini`. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

XS_LOG_TIMER

Category: Speed and Accuracy

Set this advanced option to `TRUE` to save loading and opening times in the session history log. The default value is `FALSE`. Using this advanced option you can get a quick performance overview directly from the log.

Examples of log entries:

Plug-ins loaded in 1233ms.

Plug-in dialogs loaded in 1235ms.

Opening model...

.Inp files loaded in 355ms.

Model db read in 3467ms.

Searchtree intialized in 10400ms.

Model opened in 354258743ms.

This advanced option is system specific and is read from environment files. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

XS_LOGPATH

Category

This advanced option must be set in an initialization (.ini) file.

This advanced option points to the folder that contains the Tekla Structures log file.

This advanced option is system specific and is read from environment files. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

XS_LONGHOLE_MARK_STRING_FOR_SIZE

Category

Marking: bolts

Use to define the contents of the **Size** element in slotted hole marks. For example, enter `%BOLT_NUMBER%*D%HOLE.DIAMETER%(%HOLE.DIAMETER +LONG_HOLE_X%x%HOLE.DIAMETER+LONG_HOLE_Y%).`

This advanced option is only used when there is a longhole.

The advanced options `XS_SHOP_LONGHOLE_MARK_STRING_FOR_SIZE` and `XS_SITE_LONGHOLE_MARK_STRING_FOR_SIZE` override this setting.

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)
- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD
- BOLT_MATERIAL

- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK
- BOLT_SHORT_NAME
- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SITE_LONGHOLE_MARK_STRING_FOR_SIZE (page 435)

XS_SHOP_LONGHOLE_MARK_STRING_FOR_SIZE (page 413)

XS_LONGHOLE_MARK_STRING_FOR_SIZE_IN_GA

Category

Marking: Bolts

Use to define the contents of the size element in longhole marks in general arrangement drawings. If you have not set the advanced options `XS_SHOP_LONGHOLE_MARK_STRING_FOR_SIZE_IN_GA` or `XS_SITE_LONGHOLE_MARK_STRING_FOR_SIZE_IN_GA`, then this advanced option is used.

This advanced option is only used when there is a longhole.

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)
- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD

- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK
- BOLT_SHORT_NAME
- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SHOP_LONGHOLE_MARK_STRING_FOR_SIZE_IN_GA (page 414)
XS_SITE_LONGHOLE_MARK_STRING_FOR_SIZE_IN_GA (page 436)

1.13 Advanced options - M

XS_MACRO_DIRECTORY

Category: File Locations

Specifies a global and a local folder for recorded macro files. Macro files are usually language and environment dependent and will not run in any other environment or language. This advanced option is system-specific.

Use a semicolon (;) as a separator. Do not define more than two macro folders.

First define the global folder, and then the local folder, for example:

```
set XS_MACRO_DIRECTORY=%XSDATADIR%environments\common\macros;%XSDATADIR
%environments\uk\General\user-macros
```

This advanced option is by default set to `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\macros`.

NOTE Do not change the global folder. If necessary, you may change the local folder.

When you click the **Access advanced features** button in **Applications & components** , you can select whether to create a global or local macro by selecting **New macro** --> **Local** or **> Global**. The option **Local** is not displayed, if you have not specified the folder.

NOTE The macro folder must contain subfolders `modeling` and `drawings`.

XS_MACRO_ENABLE_TIMESTAMP

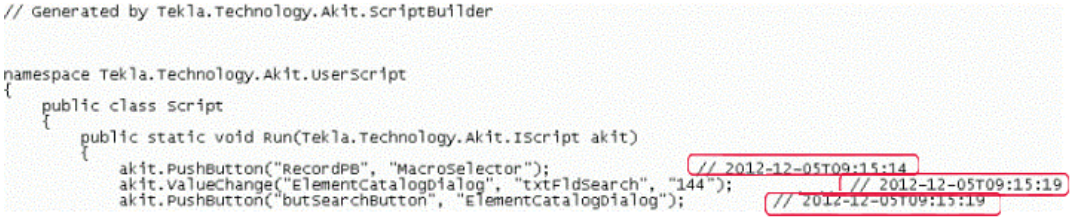
Category

Category: Modeling Properties

Set this advanced option to `TRUE` to examine the time spent on different tasks while recording macros. This advanced option is system-specific.

The default value is `FALSE`. If you change the value, you need to reopen the model to activate the new setting.

The timestamps are in the `.cs` file of the macro in question, which is located in `../environments/common/macros` folder. The macro is saved in the `drawings` or `modeling` folders, depending on in which mode the macro was recorded.



XS_MACRO_LOG

This advanced option must be set in an initialization (.ini) file.

By default Tekla Structures macro output is displayed in the terminal window. Set this advanced option to a file name to save this output to a file.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_MACRO_REFERENCES

Category: **Modeling Properties**

NOTE This advanced option is obsolete, and not used anymore by the macro engine. For macro scripting, use "#pragma reference" instead, either

with absolute path, relative path, environment variables, or in the model folder.

This advanced option defines a path to an additional library that is used when a macro is compiled. The default path is:

```
;System.Windows.Forms;Tekla.Technology.Scripting;Tekla.Structures;Tekla.Structures.Model;Tekla.Structures.Drawing;MacroSelector;System.Drawing;System.Data;System.Xml;Tekla.DataSharing.CacheServiceClient;Tekla.DataSharing.SharedPublic.
```

This advanced option is system-specific.

XS_MAGNETIC_PLANE_OFFSET

Category

Components

Use to adjust the magnetic distance of magnetic planes. By default the distance is `0.2` mm.

Setting this advanced option does not affect magnetic construction lines.

This advanced option is model specific and the setting is saved in the options database.

XS_MARK_ALL_BOLT_GROUPS_SEPARATELY

Category

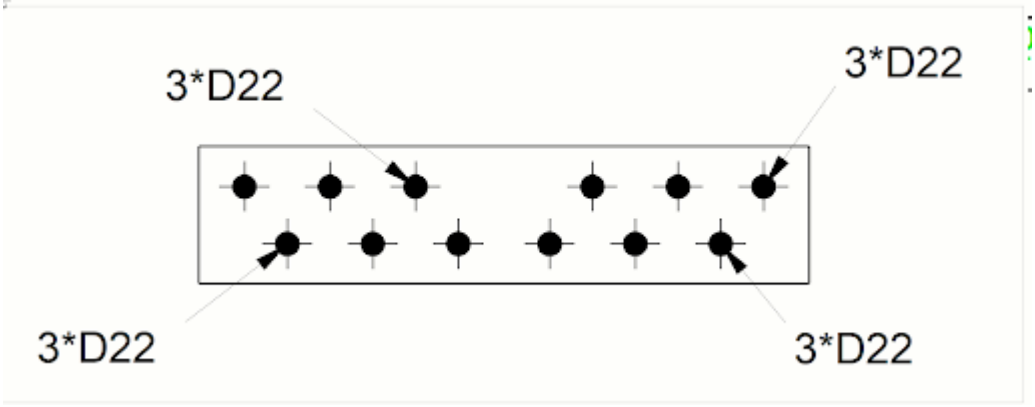
Marking: Bolts

Set this advanced option to `TRUE` to prevent Tekla Structures from combining bolt group marks. By default Tekla Structures combines bolt group marks (`FALSE`).

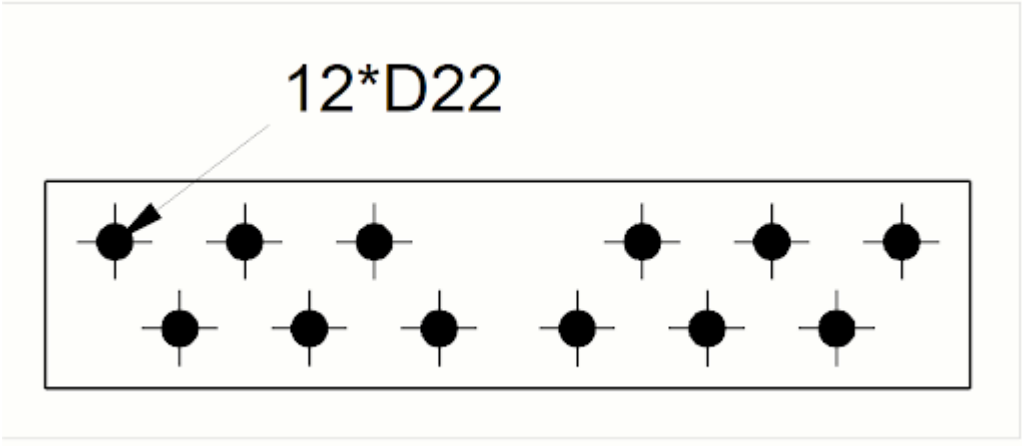
This advanced option is model specific and the setting is saved in the options database.

Example

In the image below, this advanced option is set to `TRUE`.



In the image below, `FALSE` is used.



XS_MARK_ELEMENT_SPACE_FACTOR

Category

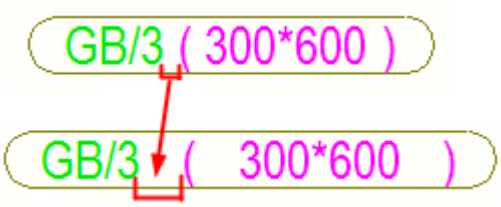
Marking: General

By default Tekla Structures leaves a space 0.3*text height between mark elements. Use this advanced option to change the default value.

This advanced option is model specific and the setting is saved in the options database.

Example

In the example below, the default value 0.3 was changed to 1.



XS_MARK_FONT

Category

Drawing Properties

You can set the mark text font (for part marks etc.) by giving the name of the font to this advanced option. The default value is Arial. If no font is given, Tekla Structures uses the default font defined for `XS_DEFAULT_FONT`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_DEFAULT_FONT (page 144)

XS_MARK_INTELLIGENT_POST_FREEPLACE_NEARBY

Category:**Marking: General**

If you set the advanced option `XS_MARK_INTELLIGENT_POST_FREEPLACE_NEARBY` to `TRUE` (default), Tekla Structures first places the marks in drawings avoiding crossing leader lines, and after that runs the place nearby command, which ensures that the mark locations follow the protection settings. If you set this advanced option to `FALSE`, the crossing mark check is performed, but the place nearby command is not run, so some of the protection settings may not be followed.

Note that you need to set the advanced option `XS_TRY_TO_KEEP_LOCATION_IN_FREEPLACING` to `TRUE` (Default) for the advanced option `XS_MARK_INTELLIGENT_POST_FREEPLACE_NEARBY` to work.

This advanced option is model specific and the setting is saved in the options database.

See also

Placement properties for marks, dimensions, notes, texts and symbols in drawings (page 784)

XS_MARK_INTELLIGENT_PLACING (page 325)
XS_TRY_TO_KEEP_LOCATION_IN_FREEPLACING (page 450)

XS_MARK_LEADER_LINE_ARROW_HEIGHT

Category

Marking: General

Height of arrow head in mark leader line. `1` is the default value. For example, the standard AutoCAD leader lide arrow height is `0.67`.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_MARK_LEADER_LINE_ARROW_LENGTH

Category

Marking: general

Length of arrow head in mark leader line. The default is `2.5`.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_MARK_LEADER_LINE_EXTENSION_LENGTH

Category

Marking: General

Use to define the length of the leader line extension. The extension is placed before the start of a text string. Give the length in millimeters. The default value is 0.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

See also

XS_MARK_LEADER_LINE_LENGTH_FOR_PERPENDICULAR

Category: **Marking: General**

Use this advanced option to control the length of the perpendicular leader lines of rebar group marks. The default value is 0.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

See also

Mark properties - Content, General, Merging and Appearance tabs (page 734)

XS_MARK_LEADER_LINE_POSITION_TYPE_FOR_NO_FRAME

Category

Marking: General

Use to define the leader line position of a leader line pointing to a mark:

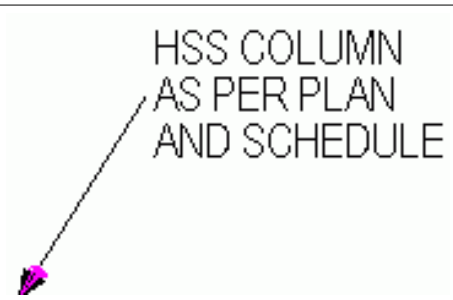
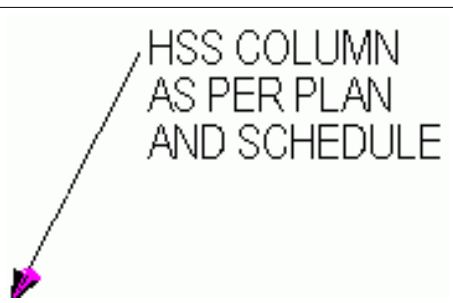
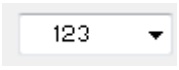
- without a mark frame (mark frame deleted from the **General** page of the mark properties dialog box)
- without a mark frame but with a mark element frame (mark frame deleted from the **General** page, and element frame selected on the **Content** page of the mark properties dialog box).

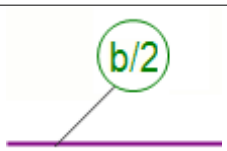
The default value is 0.

TIP You may want to keep the default value 0 for the advanced option `XS_MARK_LEADER_LINE_EXTENSION_LENGTH`, when using `XS_MARK_LEADER_LINE_POSITION_TYPE_FOR_NO_FRAME`.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

Example

Position	Appearance	Set the advanced option to
Nearest corner.		0
In the middle of the text area.		1
1/3 from the top of the text area.		2
In the middle of the first row of text.		3
The leader line connects to the frame around the mark element (not to the frame around the whole mark). The mark frame has been deleted in the mark properties. 		4

Position	Appearance	Set the advanced option to
		

See also

XS_MARK_LEADER_LINE_POSITION_TYPE_FOR_RECTANGULAR_FRAME (page 308)

XS_MARK_LEADER_LINE_EXTENSION_LENGTH (page 305)

XS_MARK_LEADER_LINE_POSITION_TYPE_FOR_RECTANGULAR_FRAME

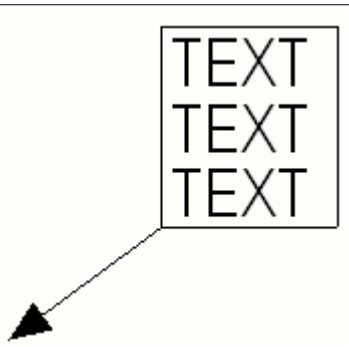
Category

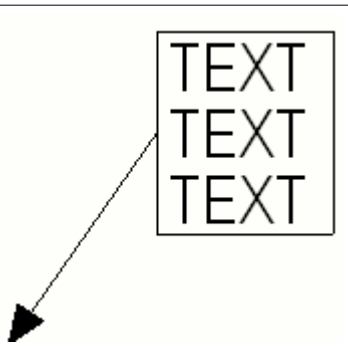
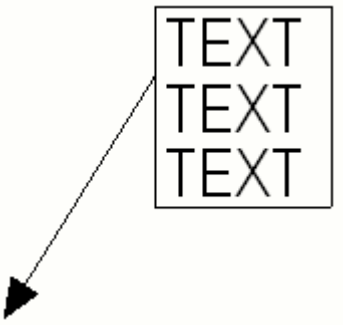
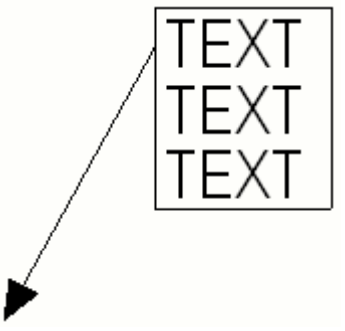
Marking: General

Use to define the leader line position for a leader line with a rectangular frame. The default value is 0.

This advanced option is model specific and the setting is saved in the options database.

Example

Position	Appearance	Set the advanced option to
Nearest corner.		0

Position	Appearance	Set the advanced option to
In the middle of the text area.		1
1/3 from the top of the text area.		2
In the middle of the firs row of text.		3

See also

XS_MARK_LEADER_LINE_POSITION_TYPE_FOR_NO_FRAME (page 306)

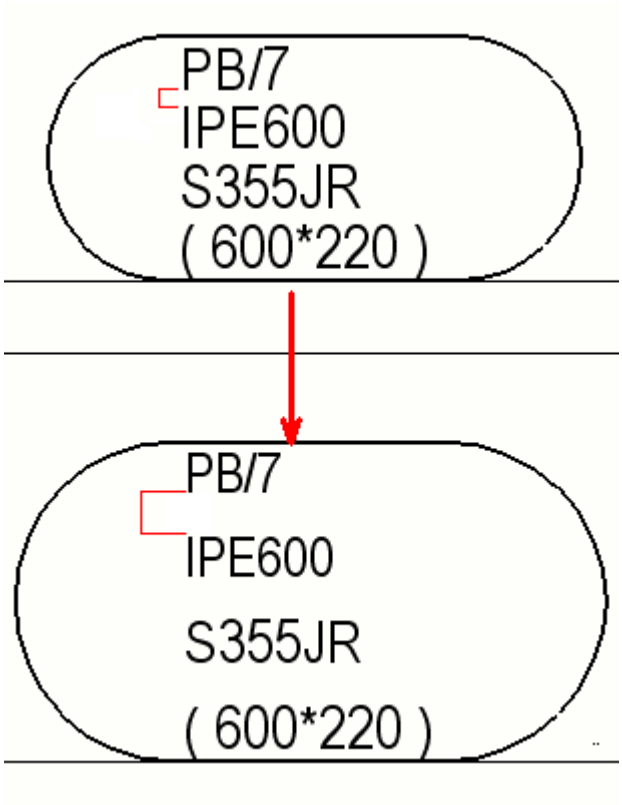
XS_MARK_LINE_SPACE_FACTOR

Category: Marking: General

By default ,Tekla Structures leaves a space of 0.3*text height between the lines in multi-line marks, for example, part, bolt, and connection marks. Use this advanced option to change the default value. This advanced option also affects the space between the lines in normal multi-line text objects.

This advanced option is model specific and the setting is saved in the options database.

In the example below, the value was changed from 0.3 to 1.



XS_MARK_PLACING_ANGLE_CLOSE_TO_45_DEGREES

Category: Marking: General

To place marks with leader lines in a 45-degree angle if allowed by the protection settings, ensure that the advanced option `XS_MARK_PLACING_ANGLE_CLOSE_TO_45_DEGREES` is set to `TRUE`. `TRUE` is the default value.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_MARK_INTELLIGENT_PLACING (page 325)

XS_MARK_TEXT_FRAME_BOX_HEIGHT_FACTOR

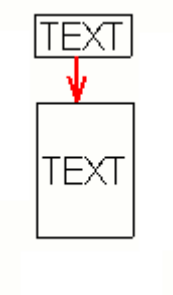
Category

Marking: General

By default Tekla Structures leaves a space of 0.5*text height between the text and the frame around the text. Use this advanced option to change the default value.

This advanced option is model specific and the setting is saved in the options database.

In the example below, the value was changed from 0.5 to 2.



XS_MATERIAL_SYMBOL_REPRESENTATION_FILE

Category

Drawing properties

Enter the folder path and filename of the file containing the user-defined material symbols, for example `material_symbol_table.txt`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_MAX_ANGLE_BETWEEN_SKEWED_END_PLATE_AND_BEAM_END

Category

Drawing Properties

Use to produce section views and dimensions of slightly skewed end plates. An end plate can be sloping or skewed to such a small degree that it is unnecessary to bevel cut the end of the main part. If the end plate is not

dimensioned in the section view, you need to set a limit for the end plate angle.

Tekla Structures dimensions any end plate skewed less than this value in section views. Larger angle dimensions do not appear in the section view. Set the angle between the skewed end plate and beam in degrees. The default angle is 0.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_MAX_ANGLE_TOLERANCE_BETWEEN_COMPLEX_MAIN_PARTS

Category

Dimensioning: Parts

Use to define the maximum angle range (0...1) within which Tekla Structures will dimension non-parallel parts as one. The default value is `0.01`.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_MAX_AUTOMATIC_RADIUS_DIMENSION

Category

Dimensioning: Parts

Defines the maximum radius when you use automatic radius in single part drawings. The default value is 5000. Tekla Structures will display radii smaller than the value you enter in drawings.

This advanced option is model specific and the setting is saved in the options database.

XS_MAX_DECIMALS_IN_PROFILE_NAME

Category

Profiles

Use this advanced option to control the number of decimals in profile names in NC file headers. The default value is 1.

NOTE This advanced option only affect plates created directly in the model. The advanced option XS_PLATE_ROUNDING_DECIMALS (page 344) affects plates created by components.

This advanced option works on plates only if you set the advanced option XS_USE_NEW_PLATE_DESIGNATION (page 473) to `TRUE` on the **Plate Work** page in the **Advanced Options** dialog box.

XS_MAX_DEVIATION_FOR_CURVED_PART_EDGES

Category

Concrete Detailing

When you camber a part, Tekla Structures calculates all the part vertices to be on a circle arc, but all span edges between two vertices are approximations of the arc. Use this advanced option to limit the maximum distance by which the edge can deviate from the arc.

Define the value in millimeters. The default is `2.0`. The minimum value is `0.1`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_WARP_MAX_ANGLE_BETWEEN_CS (page 500)

XS_MAX_FRACTIONS_IN_MODEL_DIMENSION

Category

Imperial Units

Defines the measuring accuracy in models in the US environment Imperial role. For example, this accuracy affects the imperial measurements shown with the **Measure** tool. You can enter any number, but you should use numbers such as 8, 16, 32, 64, 128 and 256. The default value is `16`.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

Example

To use the accuracy of 1/32, set this advanced option to `32`.

XS_MAXIMUM_NUMBER_OF_PLANES_TO_NAME

Category

Modeling Properties

Use to define how many planes Tekla Structures names when the planes are used, for example, in defining distance variables. The default value is 400.

When the maximum number of planes is reached, Tekla Structures stops naming the planes and uses the name **Undefined plane** for the rest of the planes.

This advanced option is model specific and the setting is saved in the options database.

XS_MAX_MERGE_DISTANCE_IN_HORIZONTAL

Category

Marking: Parts

Defines the maximum horizontal distance within which identical reinforcement get merged marks. This advanced option only affects merging of reinforcing bar marks pointing to individual reinforcing bars, not marks pointing to a group of reinforcing bars, or marks within a group of reinforcing bars. The default value is 600 mm.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_MAX_MERGE_DISTANCE_IN_VERTICAL (page 314)

XS_MAX_MERGE_DISTANCE_IN_VERTICAL

Category

Marking: Parts

Defines the maximum vertical distance within which identical reinforcement get merged marks. This advanced option only affects merging of reinforcing bar marks pointing to individual reinforcing bars, not marks pointing to a

group of reinforcing bars, or marks within a group of reinforcing bars.The default value is 600 mm.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_MAX_MERGE_DISTANCE_IN_HORIZONTAL (page 314)

XS_MAX_SPACE_BETWEEN_COMPLEX_ASSEMBLY_PARALLEL_PARTS

Category

Dimensioning: Parts

Use to define the maximum distance allowed between parallel parts for Tekla Structures to dimension them as one. The default value is `1000`.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_MDIBASICVIEWPARENT

Category

Model View

Set this advanced option to `TRUE` (default) to allow connection or default view windows to be moved within the Tekla Structures window.

Set this advanced option to `FALSE` to allow connection or default view windows to be moved anywhere on the Windows desktop.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_MDIVIEWPARENT (page 315)

XS_MDIVIEWPARENT

Category

Model View

Set this advanced option to `TRUE` to allow view windows to be moved only within the Tekla Structures window.

Set this advanced option to `FALSE` to allow view windows to be moved anywhere on the Windows desktop. This gives you more workspace on the screen, because view windows move to the front when you click on them and open dialog boxes remain behind the views. It also allows you to enlarge Tekla Structures windows to fill the entire desktop.

The default value is `TRUE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

NOTE This advanced option also affects drawing windows. Use the advanced options `XS_MDIZOOMPARENT` and `XS_MDIBASICVIEWPARENT` to control connection and default views and zoom views .

See also

XS_MDIZOOMPARENT (page 316)

XS_MDIBASICVIEWPARENT (page 315)

XS_MDIZOOMPARENT

Category

Model View

This option affects zoom windows, which can be created through the quick launch when editing drawings.

Set this advanced option to `TRUE` to allow zoom windows to be moved only within the Tekla Structures window.

Set this advanced option to `FALSE` (default) to allow zoom windows of views to be moved anywhere on the Windows desktop.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_MESSAGES

This advanced option is deprecated, but still supported in Tekla Structures version 2022 and older Tekla Structures versions. Instead of this advanced option, we recommend that you use XS_MESSAGES_PATH (page 317), which allows you to define several message file locations.

This advanced option must be set in an initialization (.ini) file.

Use to define the location of the message files. All message files have the file name extension `ail`.

This advanced option is system specific and is read from `teklastructures.ini`. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

See also

XS_MESSAGES_PATH

This advanced option must be set in an initialization (.ini) file.

Use to define the locations of the message files.

This advanced option is system specific and is read from `teklastructures.ini`. It can also be set locally, see your environment `.ini` file (`env_<environment_name>.ini`). Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

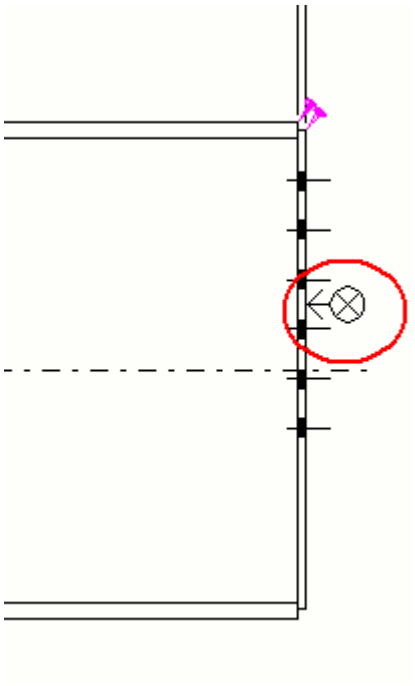
If you are an administrator, we recommend that you use the environment file for your `.ail` files.

XS_MIN_DISTANCE_FOR_CONNECTING_SIDE_MARK

Category

Marking: General

Use this advanced option to set the minimum distance of the connecting part from the main part so that when the distance is larger than the value you enter, Tekla Structures draws a connecting side mark to show that there is a part farther away from the main part that is connected to the part. When the distance is smaller than the value you enter, no mark is drawn. The default value is 300 mm.



This advanced option is model specific and the setting is saved in the options database.

See also

XS_MIN_MERGE_PART_COUNT

Category

Marking: Parts

Use to define the minimum number of identical parts whose marks to merge. The default value is 2.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_MIN_NUMBER_OF_ASSEMBLY_MULTI_CHARACTERS

Category: Numbering

Use to set the minimum number of characters in assembly multinumbers.

This advanced option is model specific and the setting is saved in the options database.

Example

If you set this advanced option to 3, the multinumber format is 101AAA.

See also

XS_USE_MULTI_NUMBERING_FOR (page 472)

XS_VALID_CHARS_FOR_ASSEMBLY_MULTI_NUMBERS (page 491)

XS_ASSEMBLY_MULTI_NUMBER_FORMAT_STRING (page 85)

XS_MIN_NUMBER_OF_PART_MULTI_CHARACTERS (page 319)

XS_MIN_NUMBER_OF_PART_MULTI_CHARACTERS

Category: Numbering

Use to define the minimum number of characters in part multinumbers.

This advanced option is model specific and the setting is saved in the options database.

Example

If you set this advanced option to 3, the multinumber format is 101aaa.

See also

XS_USE_MULTI_NUMBERING_FOR (page 472)

XS_VALID_CHARS_FOR_PART_MULTI_NUMBERS (page 492)

XS_PART_MULTI_NUMBER_FORMAT_STRING (page 340)

XS_MIN_NUMBER_OF_ASSEMBLY_MULTI_CHARACTERS (page 318)

XS_MIN_WELD_LINE_LENGTH

Category: Welds

Defines the minimum length of the weld mark reference line. If symbols and other data exceed the minimum length of weld mark reference line, weld mark

reference line extends to contain all symbols and data. Enter the value in millimeters.

This advanced option is model specific and the setting is saved in the options database.

XS_MIS_FILE_DIRECTORY

Category: CNC

Points to the folder where NC and MIS files are created. The default is the current model folder.

This advanced option is model specific and the setting is saved in the options database.

The following table shows where the NC files are created with different values entered in the **File location** box in the **NC File Settings** dialog box, and `XS_MIS_FILE_DIRECTORY` is set to `C:\NC`:

Path in File location	NC files are created in
empty	`C:\NC\ModelName`
`.\`	`C:\NC\ModelName`
`.\MyFiles`	`C:\NC\ModelName\MyFiles`

XS_MIS_SEQUENCE

Category

Export

Use to enable sequences in EJE and KISS file type MIS exports.

Define which part property is used as the sequence information. The options are:

- `CLASS`
- `PHASE_NUMBER` (default)
- `PHASE_NAME`
- `UDA:USER_PHASE`

NOTE The maximum lengths of the sequence information fields are 10 and 4 characters in KISS and EJE file types, respectively. Do not use long

names for phase names or user phases, if you are using this information as MIS Sequence.

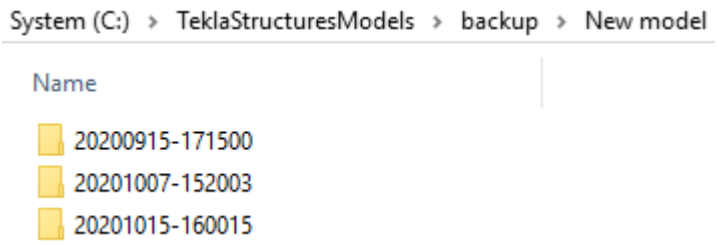
This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_MODEL_BACKUP_DIRECTORY

Category: File Locations

Enter the path to the folder where you want to store the backup copies of Tekla Structures model files. You can enter either an absolute path or a relative path. For example, you can enter a path to a mapped network drive. Model files are often large, so you should ensure that you enter a path to a location that has enough disk space.

Tekla Structures will create sub-folders for each Tekla Structures model under the folder path that you enter. Each one of these model folders will contain sub-folders for each time a backup copy of the model has been saved. These sub-folders are named by the times when backup copies have been created in the `YYYYMMDD-HHMMSS` format.



The default folder for backup copies is `..\TeklaStructuresModels\backup\`. If you do not enter another path, the default folder is used.

Note that the backup folder cannot be located directly under the actual model folder, because it would cause an infinite loop. If you define the backup folder to be in the model folder, Tekla Structures will ignore the path and use the default folder instead.

If Tekla Structures cannot find the folder path you entered, the status bar will tell you that creating a backup file has failed.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_MODEL_IMPORT_LOCK_OBJECTS

Category: **Import**

Set this advanced option to `TRUE` to lock all the imported objects. When you do this, the user-defined attribute **Locked** is automatically set to **Yes** in the user-defined attributes dialog box of the imported objects. `TRUE` is the default value.

This advanced option is model specific and the setting is saved in the options database.

XS_MODEL_PREFIX_INFLUENCES_MULTI_NUMBERING_FOR

Category: Numbering

Use to define whether part and assembly number prefixes affect the numbering of parts and assemblies in multidrawings. The options are `NONE`, `ASSEMBLIES`, `PARTS` and `ASSEMBLIES_AND_PARTS`. The value field is empty by default.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_USE_MULTI_NUMBERING_FOR (page 472)

XS_USE_MODEL_PREFIX_IN_MULTI_NUMBERS_FOR (page 471)

XS_MODEL_TEMPLATE_DIRECTORY

Category: File Locations

Enter the path to the folder where Tekla Structures saves model templates. Enter only one folder path. Only the templates saved in this folder are listed in the **Model template** list in the **New** dialog box.

For example, you can set this advanced option to point to the same location as XS_FIRM (page 260).

By default, the model template folder is saved in your environment folder, under `..ProgramData\Trimble\Tekla Structures\<version>\environments\<your environment>\`. The exact folder location may vary depending on your environment and role.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local`

`\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_MULTIDRAWING_KEEP_OBSOLETE_DRAWINGS

Category

Drawing Properties

Use this advanced option for controlling the views and multinumbers of deleted assemblies in multidrawings. Set this advanced option to `TRUE` to preserve the views of deleted parts and assemblies and to reserve the multinumbers of the deleted parts and assemblies. Set it to `FALSE` to reuse the multinumbers of the deleted assemblies and to delete the views. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_MULTIDRAWING_REMOVE_VIEW_LABEL_GAP

Category

Drawing Properties

Set this advanced option to `TRUE` to remove unnecessary space between the drawing view label and the drawing view in multidrawings. If you do not want to do this, use `FALSE` (default).

This advanced option is model specific and the setting is saved in the options database.

XS_MULTI_DRAWING_VIEW_PLACING_TRIAL_NUMBER

Category

This advanced option must be set in an initialization (.ini) file.

Use to define the number of attempts to place the drawing views into a multidrawing. Enter an integer between `1` and `500`. The advanced option is set to `500` by default.

XS_MULTI_DRAWING_VIEW_TITLE

Category

Drawing properties

Use to define a title for a multidrawing view in a multidrawing. Arbitrary strings and switches (`BASE_NAME` and `NAME`) can be used for defining the title.

By default the title is defined as follows:

```
Drawing %DRAWING_BASE_NAME%
```

This advanced option is model specific and the setting is saved in the options database.

XS_MULTI_NUMBERING_INCLUDE_ASSEMBLY_PARTS

Category: Numbering

Set this advanced option to `TRUE` to include single parts in multinumbering. If this advanced option is set to `FALSE`, single parts get multinumbers only if they are included in an assembly drawing.

The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_USE_MULTI_NUMBERING_FOR (page 472)

XS_MULTIPLIER_SEPARATOR_FOR_MERGED_PART_MARK

Category

Marking: Parts

Use to define the separator in merged part marks. The default value is x.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_MULTIUSER_SAVE_REOPEN_DISABLE_COMPACTION

This advanced option must be set in an initialization (.ini) file. We recommend that you use the `options.ini` file in the model folder.

Use this advanced option only if you frequently get the error `The command could not be completed, restart Tekla Structures and try again.` during saving a multi-user model. Set the value to `TRUE` to prevent the error, and reopen the model. The default value is `FALSE`.

1.14 Advanced options - N

XS_MARK_INTELLIGENT_PLACING

Category: Marking: General

Marks are automatically placed according to the mark placing algorithm if the advanced option `XS_MARK_INTELLIGENT_PLACING` is set to `TRUE` (default). The mark placement algorithm tries to avoid crossing leader lines, and also places marks otherwise more clearly.

If you select all four corner points in the **Part Mark Placing** dialog box, the automatic mark placement will be used, and Tekla Structures attempts to find the closest quarter with empty space.

The new algorithm applies to all marks and associative notes, except weld marks.

If `XS_MARK_INTELLIGENT_PLACING` is set to `FALSE`, you can still arrange marks and notes using the following commands:

- You can arrange the selected marks and notes so that you avoid crossing leader lines by using the **Arrange marks** command on the ribbon. The result also depends on the defined protection settings and search margins.
- With another ribbon command **Align marks** --> **Auto-align marks** you can align the selected marks and associative notes that are close by stacking the marks and placing them evenly.
- You can also align selected marks around a point using the command **Align marks** --> **Align to a point**. This command tries to avoid crossing leader lines.
- These commands can also be started from **Quick Launch**.

This advanced option is model specific and the setting is saved in the options database.

See also

Placement properties for marks, dimensions, notes, texts and symbols in drawings (page 784)

XS_NEIGHBOUR_PART_SKEW_LIMIT

Category

Drawing Properties

Tekla Structures considers neighboring parts as skewed if the product of the vector multiplication (part axis) (any of the coordinate axis) is less than 1-`XS_NEIGHBOUR_PART_SKEW_LIMIT`. Enter the limit as a floating value, for example `0.1` (default).

This advanced option is model specific and the setting is saved in the options database.

XS_NO_AUTO_DISPLAY_VIEWS

Category

Model View

Set this advance option to `TRUE` to prevent Tekla Structures from displaying views automatically when you start the program. Set it to `FALSE` (default) to display the views automatically.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_NO_BOLT_ANGLE_DIMENSIONS

Category

Dimensioning: Bolts

To create angle dimensions for bolts, set this advanced option to `FALSE`. To not create angle dimensions for bolts, select `TRUE` (default).

This advanced option is model specific and the setting is saved in the options database.

XS_NO_CHAMFERS_IN_EXACT_MODE

Category

Model view

Set this advanced option to `TRUE` to prevent Tekla Structures from creating chamfers in exact mode. To create the chamfers in exact mode, set it to `FALSE`.

By default chamfers are created. This advanced option only affects wire frame views.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_NO_END_VIEWS_TO_INCLUDED_SINGLE_DRAWINGS

Category

Single Part View in Assembly Drawing

Set this advanced option to `TRUE` (default) to prevent Tekla Structures from creating end views when you create an assembly drawing and choose to include single-part drawings. If you do not want to do this, set it to `FALSE`.

If you do not enter a value, Tekla Structures creates end views based on the properties of the single-part drawings.

This advanced option is model specific and the setting is saved in the options database.

XS_NO_RELATIVE_SHAPE_DIMENSIONS

Category

Dimensioning: Parts

Set this advanced option to `TRUE` to force shape dimensions to be the same as the dimension type you select.

By default, automatic shape dimensions are always relative regardless of the selected dimension type.

This advanced option does not affect single-part drawings. To affect single part drawings, use the advanced option XS_SINGLE_NO_RELATIVE_SHAPE_DIMENSIONS.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SINGLE_NO_RELATIVE_SHAPE_DIMENSIONS (page 426)

XS_NO_UNFOLDING_LINES_TO_DRAWINGS

Category

Dimensioning: Unfolding

Use to define whether unfolding lines are shown in drawings. When you set the advanced option to TRUE, the unfolding lines are not shown. The default is FALSE.

This advanced option is model specific and the setting is saved in the options database.

XS_NO_SINGLE_PART_DRAWINGS_FOR

Category

Single Part View in Assembly Drawing

Set as follows to prevent Tekla Structures producing single part drawings for assemblies containing one part:

XS_NO_SINGLE_PART_DRAWINGS_FOR=LOOSE_PARTS

This advanced option is model specific and the setting is saved in the options database.

XS_NORTH_MARK_SCALE

Category

Marking: Parts

Use this advanced option to define the scale of the north mark symbol. By default, North marks have a scale of 1:1. You can also create a larger symbol for north marks in the Symbol editor.

This advanced option is model specific and the setting is saved in the options database.

XS_NORTH_MARK_SYMBOL

Category: Marking: Parts

By default the north mark symbol is number 32 in the symbol file `xsteel.sym`. Use this advanced option to change the symbol.

This advanced option is model specific and the setting is saved in the options database.

See also

Show orientation marks (north marks)

XS_NSFS_POSTFIX_FOR_MERGED_PART_MARK

Category

Marking: Parts

Use to define the postfix in merged part marks. This postfix is visible for identical parts on both side of a main part. The default value is BS.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_NSFS_TEXT_POSITION_IN_PART_MARK

Category

Marking: parts

Defines which element the NS, FS or BS text follows in merged part marks.

The default is 23, which means that the text is located after part position. If the type that is defined by the advanced option cannot be found in the mark at all, the text is located at end of the mark. To force the text to always appear at the end of the mark, use -1.

This advanced option is model specific and the setting is saved in the options database.

For more information about the automatic merging of the part marks, see .

The following list contains the corresponding integer values and elements in the mark:

TEXT = 1

LINE FEED = 2

SYMBOL = 3

FRAME START = 4

FRAME END = 5

MATERIAL = 10

USER DEFINED ATTRIBUTE = 16

ASSEMBLY_POSITION = 22

PART_POSITION = 23

PROFILE = 24

NAME = 25

LENGTH = 26

CAMBER = 27

SIZE = 28

FITTINGS (NS/FS) = 29

CLASS = 38

BACK_SPACE = 46

GAGE OF OUTSTANDING LEG = 48

CENTER TO CENTER DISTANCE = 49

FACE_DIRECTION = 57

END OF Mark = -1

Example

`XS_NSFS_TEXT_POSITION_IN_PART_MARK=22`

The value 22 means after assembly position.

NOTE NS, FS and BS text itself comes from two separate places depending on if the marks are merged or not. For ordinary marks, the text comes from a file called `by_number.ail` (NS: by_number_msg_no_675, FS: by_number_msg_no_676). For merged marks, the text comes from the following advanced options:

- BS: `XS_GET_NSFS_POSTFIX_FOR_MERGED_PART_MARK`
- NS: `XS_GET_NS_POSTFIX_FOR_MERGED_PART_MARK`
- FS: `XS_GET_FS_POSTFIX_FOR_MERGED_PART_MARK`

XS_NS_POSTFIX_FOR_MERGED_PART_MARK

Category

Marking: Parts

Use to define the near side postfix in merged part marks. This postfix is visible for identical parts on near side. The default value is NS.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_NUMBERING_RESULTS_DIALOG_DISPLAY_TIME

Category

Numbering

Use to set the time frame within which Tekla Structures makes the second save when numbering when you have selected the **Synchronize with master model (save-numbering-save)** option in the **Numbering setup** dialog box.

Enter the desired time in seconds. The advanced option is set to `1500` by default.

This advanced option is model specific and the setting is saved in the options database.

See also

1.15 Advanced options - O

XS_OBJECT_SELECTION_CONFIRMATION

Category

Modeling Properties

Enter the time in milliseconds after which Tekla Structures prompts you to cancel object selection. You can cancel the object selection process if the selection takes longer than the defined time.

The default value is `5000`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_OBJECTLOCK_DEFAULT

Category

Modeling Properties

Set the default lock status for new assemblies or cast units when they are created. Additionally, when you start to share the model in Tekla Model Sharing, the default lock status is set for all assemblies and cast units that do not yet have any lock status. The locked status is shown in the **Locked** value in the **Object locks** dialog box. The value options are `ORGANIZATION` or `NO`.

This advanced option is system specific.

XS_OMIT_MARKS_OF_HIDDEN_PARTS_IN_GA_DRAWINGS

Category

Marking: Parts

Set this advanced option to `TRUE` to hide part marks of parts hidden by other parts from views in general arrangement drawings. The default value is `FALSE`.

If you have stiffeners on either side of a beam, one will be hidden by the beam in front of it. Set this advanced option to `TRUE` to prevent Tekla Structures from displaying the part mark of the hidden part.

This advanced option is model specific and the setting is saved in the options database.

XS_OMIT_MARKS_OF_PARTS_OUT_OF_VIEW_PLANE_LIMIT_ANGLE

Category

Marking: General

Use to hide the part marks for parts outside the current view plane by defining the limit for inclusion as an angle. The default value is `20.0`.

You also need to set **Parts out of view plane** option in the view level **Part mark properties** dialog box to **Not visible** to hide parts from the views that are outside the angle you specify here.

This advanced option is model specific and the setting is saved in the options database.

See also

Mark properties - Content, General, Merging and Appearance tabs (page 734)

XS_OMITTED_BOLT_ASSEMBLY_TYPE

Category: **Marking: Bolts**

Use this advanced option to filter out the given type of bolt marks when the **Ignore size** setting is set to a value in drawing bolt mark properties. The options are:

- `SITE` (default)
- `SHOP`
- `SITE_AND_SHOP`.

The default value `SITE` filters out only site bolt marks for bolts that meet the **Ignore size** value, whereas workshop bolt marks of all sizes are shown in drawings.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_OMITTED_BOLT_TYPE (page 334)

XS_OMITTED_BOLT_TYPE

Category: Marking: Bolts

Use this advanced option to define which bolt standards to omit from drawings when a size has been defined for the **Ignore size** setting in drawing bolt mark properties. Enter the name of the bolt standard, for example, `7990`. You can also use wildcards, such as * or ?. By default, no value is given, which means that no bolt standards are omitted.

Example: First define a value to **Ignore size**. To filter away all the marks of bolts of that size, and the marks of bolts of the bolt standards A325N, A325X and A325SC, set this advanced option to `A325*`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_OMITTED_BOLT_ASSEMBLY_TYPE (page 333)

XS_OMITTED_DIAMETER_TYPE (page 334)

XS_GA_OMITTED_DIAMETER_TYPE (page 265)

XS_OMITTED_DIAMETER_TYPE

Category

Marking: Bolts

Use to define the bolt diameter type of the marks to omit in drawings. The options are `HOLE` or `BOLT`.

This advanced option is used together with the drawing bolt mark property setting **Ignore size**. For example, to filter out all bolt marks of bolts that have a hole diameter 22, enter 22 as the value for **Ignore size**, and set this advanced option to `HOLE`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_OMITTED_BOLT_TYPE (page 334)

XS_OMITTED_PART_NAME_IN_AUTOCONNECTION

Category

Components

Use to filter out specific part types when you use AutoConnection.AutoConnection cannot handle brace connections when large quantities of parts are selected.

This advanced option is model specific and the setting is saved in the options database.

Example

If you want to use this advanced option to filter out brace parts, set it to `brace`. Tekla Structures does not select any parts with names containing the string “brace”.

TIP You can also do this by setting the **Selection Filter** to select all parts except for those named “brace*”.

XS_OMITTED_WELD_TYPE

Category

Welds

Use to define which weld types to omit in drawings. Enter the number of the weld type you want to omit. The default value is `10` (fillet weld). For more information about the weld types and the corresponding numbers, see List of weld types.

In addition to this advanced option there are two other settings that Tekla Structures uses to determine the welds that are shown in the drawing: `XS_WELD_FILTER_TYPE` tells if Tekla Structures should filter welds that are exactly (`EXACT`) the size or of the same size or smaller (`MIN`) than the value given in the **Weld size limit** box in the object or mark properties on the drawing view level. Tekla Structures always shows welds that have reference text.

This advanced option is model specific and the setting is saved in the options database.

Example

If you set `XS_OMITTED_WELD_TYPE` to 10, `XS_WELD_FILTER_TYPE` to `EXACT`, and **Weld size limit** to 5, Tekla Structures shows all other welds except those that are 5 mm, and those of type fillet weld (10). In this case, if you do not set

XS_WELD_FILTER_TYPE, Tekla Structures shows all welds that are bigger than 5 mm except fillet welds.

See also

XS_WELD_FILTER_TYPE (page 500)

XS_OPEN_DRAWINGS_MAXIMIZED

Category

Drawing View

Set to TRUE to maximize drawings when you open them. The default is FALSE.

This advanced option is user specific and the setting is saved in options.bin under the user folder, for example, in C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings. Restart Tekla Structures to activate the new value.

XS_ORIENTATION_MARK_DIRECTION

Category

Marking: parts

Use to define the direction of orientation marks. The options are NORTH-EAST, NORTH-WEST, SOUTH-EAST and SOUTH-WEST. NORTH-EAST is the default value.

This advanced option is model specific and the setting is saved in the options database.

See also

Show orientation marks (north marks)

XS_NORTH_MARK_SYMBOL (page 329)

XS_ORIENTATION_MARK_MOVE_DIST_FOR_BEAMS

Category

Marking: parts

Use to indicate the position of orientation marks for beams. Enter a value indicating the distance from the end of the part to the orientation mark. The default value is 300.0 mm. You can enter any value in the range 1.0 to 3000.0.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_ORIENTATION_MARK_MOVE_DIST_FOR_BEAMS_IN_GA

Category

Marking: Parts

Indicates the position of beam orientation marks in GA drawings. Enter a value indicating the distance from the end of the part to the orientation mark. The default value is 300.0 mm.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_ORIENTATION_MARK_MOVE_DIST_FOR_BEAMS (page 336)

XS_ORIENTATION_MARK_MOVE_DIST_FOR_COLUMNS

Category

Marking: parts

Use to define the placing of orientation marks for columns. The value assigned to this advanced option is the distance from the end of the part to the orientation mark. The default value for columns is 300.0 mm. Possible values range from 1.0 to 3000.0.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

See also

XS_ORIENTATION_MARK_MOVE_DIST_FOR_COLUMNS_IN_GA

Category

Marking: Parts

Defines the position of column orientation marks in GA drawings. The value assigned to this advanced option is the distance from the end of the part to the orientation mark. The default value is 300.0 mm.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

See also

XS_ORIENTATION_MARK_MOVE_DIST_FOR_COLUMNS (page 337)

1.16 Advanced options - P

XS_PARAMETRIC_PROFILE_SEPARATOR

Category

Profiles

Use this advanced option to define an additional character to separate dimensions in the name of parametric profiles. Tekla Structures always recognizes the standard separator characters X, *, -, and /. The default value is an asterisk (*).

For example, when you inquire a part with a sketched profile, the character specified by this advanced option is used as the separator.

Example

```
XS_PARAMETRIC_PROFILE_SEPARATOR=E
```

Acceptable profile names with this setting are:

1. PL500*800
2. PL500X800
3. PL500E800

Any combination of these is also accepted:
`ProfileName500*500-500*500E500` (the profile `ProfileName` should be defined).

Limitations

- Only one character is accepted as a value for this advanced option.
- You cannot use slash (/) in the US imperial environment.

See also

XS_USER_DEFINED_PARAMETRIC_PROFILE_SEPARATORS (page 490)

XS_PART_DIMENSION_PLANES_TABLE

Category

Dimensioning: Parts

Use to indicate the path to the user-defined part dimension planes table. This table defines the planes in which dimensions are created. For example, you might want Tekla Structures to dimension round bars to the middle of the profile rather than the reference line.

You can also use a file name as a value. If the value is a file name, Tekla Structures searches for the file in the model, project, firm and profile folders (in this order).

This is a system-specific advanced option.

Example

```
XS_PART_DIMENSION_PLANES_TABLE=%XS_PROFDB%
\dim_planes_table.txt
```

See also

XS_PART_MERGE_MAX_DISTANCE

Category

Marking: Parts

Use to define the maximum distance within which identical parts get merged marks. Units are in millimeters. The default value is 1200.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

See also

XS_PART_MULTI_NUMBER_FORMAT_STRING

Category: Numbering

Use to define multinumbers for single parts. Use the following options to define the contents of part marks. Use as many switches as you need, and enclose each one in percent symbols (%).

The available options are:

Option	Description
`%PART_MULTI_DRAWING_NUMBER%`	Multidrawing name.
`%PART_MULTI_DRAWING_POS%`	Position of the single-part drawing inside the multidrawing.
`%PART_PREFIX%`	Part prefix in the model.
`%PART_POS%`	Part position number in the model.
Template fields	Enter `TPL:` followed by the name of any relevant template field. Enclose each name in percent symbols (%). For example, `%TPL:PROJECT.NUMBER%`
User-defined attributes that are defined in the `objects.inp` file	Enter `UDA:` followed by the name of any relevant user-defined attribute, exactly as it appears in the `objects.inp` file. For example, `%UDA:MY_INFO_1%`

This advanced option is model specific and the setting is saved in the options database.

Example

To put the part multinumbers in the format part prefix + position on multidrawing + multidrawing name, set the advanced option as follows:

```
%PART_PREFIX%%PART_MULTI_DRAWING_POS%
%PART_MULTI_DRAWING_NUMBER%
```

See also

XS_USE_MULTI_NUMBERING_FOR (page 472)

XS_ASSEMBLY_MULTI_NUMBER_FORMAT_STRING (page 85)

XS_CAST_UNIT_MULTI_NUMBER_FORMAT_STRING (page 109)

XS_PART_POSITION_NUMBER_FORMAT_STRING

Category: Numbering

Use to have Tekla Structures use only letters in part numbers.

NOTE The advanced options XS_ASSEMBLY_POSITION_NUMBER_FORMAT_STRING (page 87) and `XS_PART_POSITION_NUMBER_FORMAT_STRING` override the advanced option `XS_SWITCH_POS_NUMBERS_FOR` (page 445). `XS_SWITCH_POS_NUMBERS_FOR` has no impact on assembly and/or part position numbers if you use `XS_ASSEMBLY_POSITION_NUMBER_FORMAT_STRING` and/or `XS_PART_POSITION_NUMBER_FORMAT_STRING`.

Enter any combination of the following options:

Option	Description
`%PART_PREFIX%`	Part prefix, defined in the part properties in the property pane.
`%PART_POS%`	Part position number, defined by the start number (from part properties in the property pane) and the final position in that numbering series.
`%PART_POS_WITH_LETTERS%`	Same as above, but with letters. Uses letters A – Z by default, but you can also define valid letters with the advanced option `XS_VALID_CHARS_FOR_PART_POSITION_NUMBERS`.

The position number/ letter switch can also include a suffix defining the minimum number of digits (or letters), for example: `%PART_POS.3%`. This example results in a first part number of 001, second 002 etc.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_VALID_CHARS_FOR_PART_POSITION_NUMBERS (page 493)

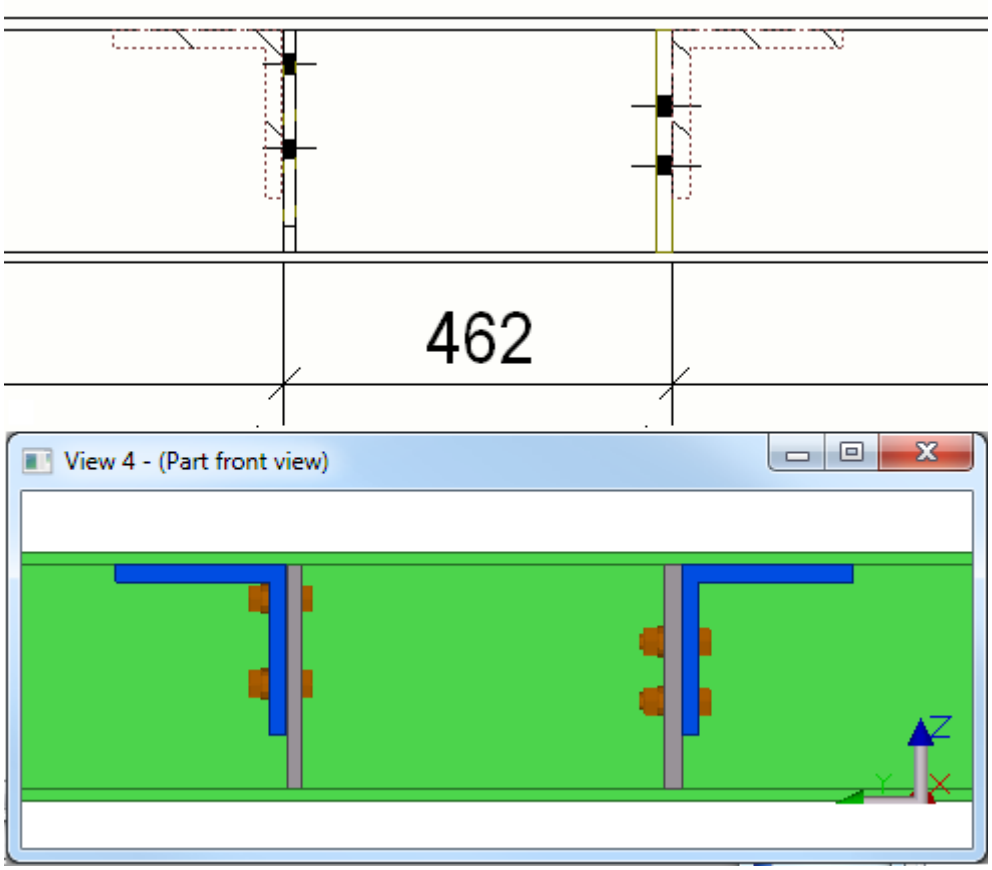
XS_PART_POSITION_TO_EDGE_NEAREST_TO_NEIGHBOUR

Category

Dimensioning: Parts

Set this advanced option to `TRUE` to dimension the position of parts to the edge that is nearest to the neighbor part. The default is `FALSE`.

In the image below, the neighbor parts in the model are blue, and the plate creation points are shown.



For more information about using advanced options when adding dimensions to plates, see .

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_PART_POSITION_TO_LEADING_EDGE

Category

Dimensioning: Parts

Set this advanced option to `TRUE` (default) to dimension the position of beams to the leading edge. If you do not want to do this, set it to `FALSE`. For this advanced option to take effect, disable the advanced option `XS_USE_PLATE_SIDE_POSITIONING`.

For column assemblies, you must also set the advanced option XS_PART_POSITION_TO_LEADING_EDGE_IN_COLUMNS_ALSO to TRUE.

For more information about using advanced options when adding dimensions to plates, see Add dimensions to plates.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_PART_POSITION_TO_LEADING_EDGE_IN_COLUMNS_ALSO (page 343)

XS_USE_PLATE_SIDE_POSITIONING (page 481)

XS_PART_POSITION_TO_LEADING_EDGE_IN_COLUMNS_ALSO

Category

Dimensioning: Parts

To dimension part position to the leading edge for column assemblies, set this advanced option to TRUE. If you do not want to do this, set it to FALSE. The default value is TRUE.

You must also set the advanced option XS_PART_POSITION_TO_LEADING_EDGE (page 342) to TRUE.

For more information about using advanced options when adding dimensions to plates, see Add dimensions to plates.

This advanced option is model specific and the setting is saved in the options database.

XS_PIXEL_TOLERANCE

Category

Model View

Use to define the snap zone of objects. Each object has a snap zone, which defines how close you need to pick to hit a position. When you pick within the snap zone of an object, Tekla Structures automatically snaps to the closest pickable point on that object. Enter the value in pixels. The default value is 10.

This advanced option is user specific and the setting is saved in options.bin under the user folder, for example, in C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings. Restart Tekla Structures to activate the new value.

XS_PLATE_ROUNDING_DECIMALS

Category

Plate Work

Use to define the maximum number of decimals in plate profile name created by **components**. The default is 1.

Note that unnecessary zeroes are always left off, for example 10.501:

- with 2 decimals is 10.5
- with 3 decimals is 10.501

This advanced option is model specific and the setting is saved in the options database.

The advanced option `XS_MAX_DECIMALS_IN_PROFILE_NAME` controls decimals of plates that are modeled directly in the model.

See also

XS_MAX_DECIMALS_IN_PROFILE_NAME (page 312)

XS_PLOT_ORIGIN_MOVE_X

Category

This advanced option must be set in an initialization (.ini) file.

Offsets the print origin in the x direction. Use if a drawing does not fit to the paper or is printed to a wrong location. Enter the value in millimeters as an integer. This advanced option affects all printers. By default, this advanced option is not set to any value.

NOTE If you set these advanced options in your initialization files, you will override the **Printer Catalog** dialog box.

See also

XS_PLOT_ORIGIN_MOVE_Y (page 344)

XS_PLOT_ORIGIN_MOVE_Y

Category

This advanced option must be set in an initialization (.ini) file.

Offsets the print origin in the y direction. Use if a drawing does not fit to the paper or is printed to a wrong location. Enter the value in millimeters as an integer. This advanced option affects all printers. By default, this advanced option is not set to any value.

NOTE If you set these advanced options in your initialization files, you will override the **Printer Catalog** dialog box.

See also

XS_PLOT_ORIGIN_MOVE_X (page 344)

XS_PLOT_VIEW_FRAMES

Category

Printing

To show drawing view frames in printed and exported drawings, set `XS_PLOT_VIEW_FRAMES` to `TRUE`. `FALSE` is the default value.

This advanced option is model specific and the setting is saved in the options database.

PML_ASSEMBLY_MARKS_IN_USE

Category

Export

Set this advanced option to `TRUE` to have Tekla Structures use assembly marks in PML exports. By default this advanced option is set to `FALSE`, which means that Tekla Structures uses part marks.

This advanced option is model specific and the setting is saved in the options database.

PML_CARDINAL_POINT_NOT_IN_USE

Category

Export

Set this advanced option to `TRUE` to prevent Tekla Structures from using cardinal points in PML exports. This means that all parts will be defined by their center lines, and their position may differ from that in the Tekla

Structures model. By default this advanced option is set to FALSE, which means that cardinal points are used in PML export. The default value is FALSE.

XS_PML_EXPORT_INCLUDE_GLOBAL_ID

Category

Export

Set this advanced option to TRUE to revert to the FrameWorksPlus ID number in PML exports. If you do not want to export the ID number, set it to FALSE. The default value is FALSE.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SDNF_IMPORT_STORE_MEMBER_NUMBER (page 399)

Export to CAD

XS_PML_EXPORT_USE_ADDITIONAL_CUT_DIST

Category

Export

Some earlier versions of Tekla Structures added 1 mm in length to fitted part ends in PML exports. Set this advanced option to TRUE to force recent versions to add the length. The default value is FALSE.

This advanced option is model specific and the setting is saved in the options database.

XS_POINT_CLOUD_CACHE_FOLDER

Category: File Locations

Use this advanced option to define the folder where point cloud data is stored. By default, the folder is %LocalAppData%\Trimble\Tekla Structures\PointClouds, for example, C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\PointClouds.

This advanced option is user specific and the setting is saved in options.bin under the user folder, for example, in C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings. Restart Tekla Structures to activate the new value.

XS_POINT_CLOUDS_WEB_CACHE

Category: File Locations

Use this advanced option to define the point cloud web streaming cache. By default, the folder is `%LocalAppData%\Trimble\Trimble Connect\Import`, for example, `C:\Users\<user>\AppData\Local\Trimble\Trimble Connect\Import`.

XS_POLYBEAM_CHORD_TOLERANCE

Category: Speed and Accuracy

Use this advanced option to define the chord tolerance for curved polybeam segments. Enter the value in millimeters. The default is `1.0`.

Re-open the model to activate the new value.

NOTE Do not change the chord tolerance settings during a project. Changing them automatically recreates the curved polybeams when you re-open the model, resulting in slightly different solid objects, which may affect numbering, or the concrete covers of rebar sets, for example.

The advanced option `XS_POLYBEAM_MAX_ANGLE_BETWEEN_CS` works as limiting factor to `XS_POLYBEAM_CHORD_TOLERANCE`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_POLYBEAM_MAX_ANGLE_BETWEEN_CS (page 347)

XS_CHORD_TOLERANCE_FOR_TUBE_SEGMENTS (page 119)

XS_POLYBEAM_MAX_ANGLE_BETWEEN_CS

Category: Speed and Accuracy

Use this advanced option to define the maximum angle between adjacent cross sections in curved polybeam segments. Enter the value in degrees. The default value is `30.0`.

This advanced option works as a limiting factor to `XS_POLYBEAM_CHORD_TOLERANCE`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_POLYBEAM_CHORD_TOLERANCE (page 347)

XS_POLYBEAM_CURVATURE_TOLERANCE

Category

Modeling Properties

Use to define the tolerance used in detecting the curvature between three points in a polybeam. The default is `2.0e-6`.

This advanced option defines the difference in dot products between two unit vectors formed by two consecutive polybeam arc chamfer handles. If the dot product differs less than this value, the curve is considered to be a straight line, and the arc chamfer is omitted.

Generally, you need to change the default value only if you are working with long, thin or very complex polybeams. Change the default value in the following situations:

- If the polybeam is only very slightly curved and it looks like a straight polybeam in the model, you need to give a smaller value, such as `2.0e-10`. A larger value makes polybeams with only a slight curvature straight.
- If the value is unnecessarily small (smaller than default value for simple polybeams), there might be performance issues.
- If the tolerance is set to a value that is too small (`< e-11`), the polybeam may break.

XS_POLYGON_CUT_EXTRA_THICKNESS

Category

Modeling properties

Use this advanced option to define the cut depth of a polygon cut, for example, to cut thick surface treatment. The default value for the cut thickness is 5.0 mm.

XS_POLYGON_PERPENDICULAR_EDGE_PREFERENCE_FACTOR

Category

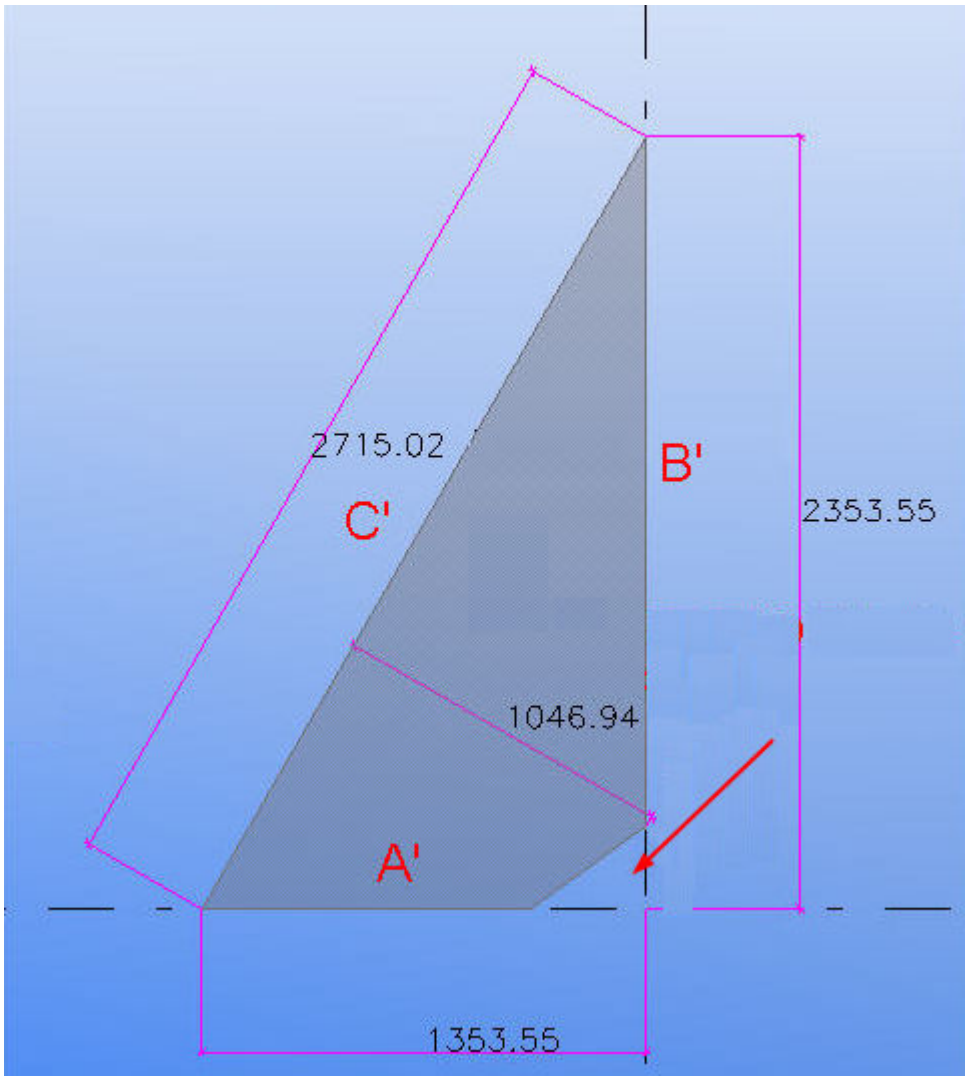
Plate Work

This advanced option is used to artificially manipulate which side of a plate is considered as the 'longest'.

The longest side of polygon plates always faces downwards in drawings, which can affect plates with perpendicular edges.

This information can then be used, for example, in changing the rotation of a plate in drawings, or when choosing which side of a plate is to be considered the 'Length' and 'Width'.

This advanced option is used for plates with perpendicular sides in drawings. This advanced option rotates plates if there is some edge which is perpendicular to the current one and it is not adjacent edge.



The default value is 1.5.

In the example above, when you set `XS_POLYGON_PERPENDICULAR_EDGE_PREFERENCE_FACTOR` to 1, all sides are multiplied by 1, and view restriction box is drawn according to the longest side. The result is that the Length = 2715.02, and Width = 1046.94.

If you set this advanced option to 10, perpendicular corner edges A' and B' are multiplied by 10. If the result is larger than the longest side C', the view restriction box is drawn using the sides A' and B'. The result would be Length = 2353.55, and Width = 1353.55.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_POLYGON_SQUARE_CORNER_PREFERENCE_FACTOR (page 350)

XS_POLYGON_SQUARE_CORNER_PREFERENCE_FACTOR

Category

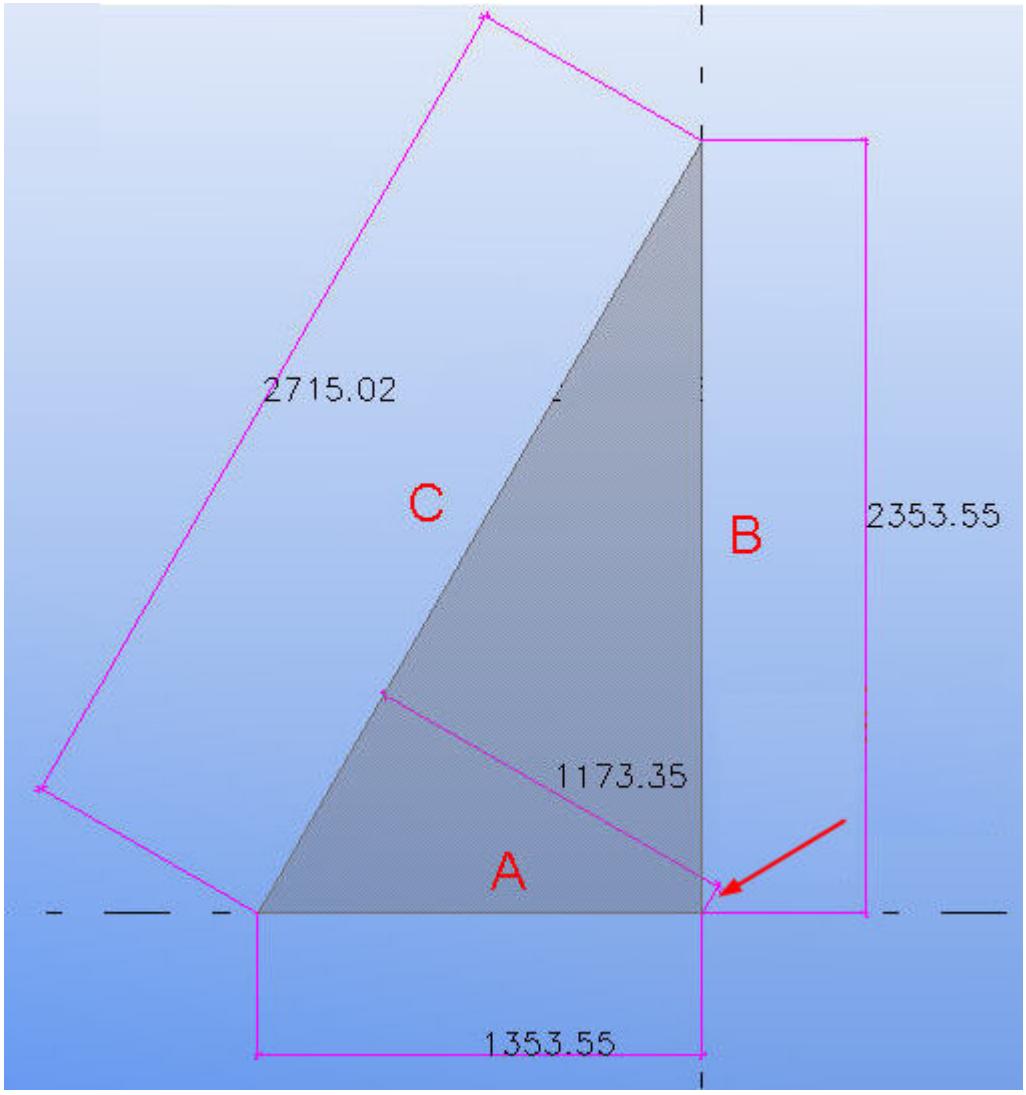
Plate Work

This advanced option is used to artificially manipulate which side of a plate is considered as the 'longest'.

In drawings, this advanced option is also used to control the rotation if right-angled plates. The longest side of polygon plates always faces downwards in drawings, which can affect right-angled plates.

This information can then be used, for example, in changing the rotation of a plate in drawings, or when choosing which side of a plate is to be considered the 'Length' and 'Width'.

This advanced option is used for plates that have two consecutive edges perpendicular to each other. When this advanced option is set to a factor, Tekla Structures multiplies the length of the side next to the right angle by that factor, making it the longest side. In reports, this side will be then considered the 'Length', and the corresponding perpendicular distance the 'Width'.



Tekla Structures still uses and displays the real dimensions of the plate.

In the example above, when you set `XS_POLYGON_SQUARE_CORNER_PREFERENCE_FACTOR` to 1, all sides are multiplied by 1, and the view restriction box is drawn according to the longest side. The result is Length = 2715.02, and Width = 1173.35.

If you set this advanced option to 10, only square corner edges A and B are multiplied by 10. If the result is larger than the largest side C, the view restriction box is drawn using the sides A and B. The result would be Length = 2353.55, and Width = 1353.55.

The default value for this advanced option is 2.0.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_POLYGON_PERPENDICULAR_EDGE_PREFERENCE_FACTOR (page 348)

XS_POP_MARK_COLOR

Category

Drawing Properties

Use to define the color of a customized pop-mark symbol that is displayed in a drawing. Enter an integer. The default value is `1` (white). Other possible values are:

Value	Pop mark color
0	Black
2	Red
3	Green
4	Blue
5	Cyan
6	Yellow
7	Magenta

This advanced option is model specific and the setting is saved in the options database.

See also

XS_POP_MARK_SYMBOL (page 353)

XS_POP_MARK_HEIGHT (page 352)

XS_POP_MARK_HEIGHT

Category

Drawing Properties

Use to define the height of a customized pop-mark symbol that is displayed in a drawing. Enter a decimal value in millimeters. The default is `2.0`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_POP_MARK_SYMBOL (page 353)

XS_POP_MARK_COLOR (page 352)

XS_POP_MARK_SYMBOL

Category

Drawing Properties

Use to define the customized pop-mark symbol to be used for pop-marks in drawings. The default is `xsteel@0`, which means that you are using symbol number `0` in the symbol file `xsteel`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_POP_MARK_HEIGHT (page 352)

XS_POP_MARK_COLOR (page 352)

XS_POSITION_DIMENSIONS_FOR_HOLES _IN_SINGLE_SECONDARY_PARTS_IN_ASSEMBLY_DRAWING

Category

Dimensioning: Bolts

To create position dimensions for holes in single secondary parts in assembly drawing, set this advanced option to TRUE. To not create position dimensions for holes, set it to `FALSE`. The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

XS_POUR_BREAK_COLOR

Category: Concrete Detailing

Use this advanced option to change the color of pour breaks in model views. Enter number as the value using the class numbers in the part property pane to indicate the color. For example, if you set this advanced option to 6, Tekla Structures will color all pour breaks yellow. The default value is 59.

In the exported IFC models, pour breaks are black.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_POUR_OBJECT_COLOR (page 354)

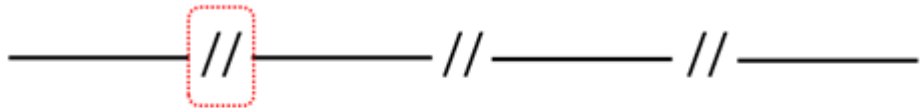
XS_INVALID_POUR_BREAK_COLOR (page 289)

XS_POUR_BREAK_SYMBOL

Category

Drawing Properties

Pour breaks are represented by a symbol in the drawings, see the image below. The symbol scale and the spacing between the symbols follows the drawing view scale automatically.



If you want to change the pour break symbol, enter a new value for this advanced option. The default value is `PourBreaks@0`. The symbol value starts with the symbol library file name and ends with the number of the symbol. The default library may contain many different pour break symbols. If you wish to use a symbol file that is not located under your environment folders, enter the complete path to the symbol file location and the symbol file name.

This advanced option is model specific and the setting is saved in the options database.

XS_POUR_OBJECT_COLOR

Category: Concrete Detailing

Use this advanced option to change the default color of pour objects in model views. Enter number as the value using the class numbers in the part property

pane to indicate the color. For example, if you set this advanced option to 6, Tekla Structures will color all pour objects yellow.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_POUR_BREAK_COLOR (page 353)

XS_PREVIEW_LIMIT

Category: **Model View**

Use this advanced option to set the limit for number of objects shown in the copy or move preview. The preview is shown in the model when you use the **Copy** or **Move** command to copy or move objects.

The default value is 1000. When the value is 0, the preview is off.

XS_PRINT_MULTISHEET_BORDER

Category

Drawing Properties

Use to define the borders that are left out from the smaller sheets when printing a drawing on multiple small sheets.

For example, to leave a 3 mm horizontal and a 5 mm vertical border, set the advanced option to 3,5.

XS_PRINT_REPORT_FONT

Category

Templates and Symbols

Define the font for printed reports. Tekla Structures uses this advanced option if you do not specify another font for printed reports in the **Print** dialog box. The default value is `Arial Narrow`. If you do not enter a font, Tekla Structures uses the default font defined for `XS_DEFAULT_FONT`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local`

`\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_PRINT_REPORT_LINE_WIDTH_LANDSCAPE

Category

Templates and symbols

Use to specify the number of characters per row in reports printed in landscape orientation. The default value is `132`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_PRINT_REPORT_LINE_WIDTH_PORTRAIT (page 356)

XS_PRINT_REPORT_PAGE_HEIGHT_LANDSCAPE (page 356)

XS_PRINT_REPORT_PAGE_HEIGHT_PORTRAIT (page 357)

XS_PRINT_REPORT_LINE_WIDTH_PORTRAIT

Category

Templates and symbols

Use to specify the number of characters per row in reports printed in portrait orientation. the default value is `80`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_PRINT_REPORT_LINE_WIDTH_LANDSCAPE (page 356)

XS_PRINT_REPORT_PAGE_HEIGHT_LANDSCAPE (page 356)

XS_PRINT_REPORT_PAGE_HEIGHT_PORTRAIT (page 357)

XS_PRINT_REPORT_PAGE_HEIGHT_LANDSCAPE

Category

Templates and symbols

Use to specify the number of rows in printed reports with landscape orientation. The default value is `42`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_PRINT_REPORT_LINE_WIDTH_LANDSCAPE (page 356)

XS_PRINT_REPORT_LINE_WIDTH_PORTRAIT (page 356)

XS_PRINT_REPORT_PAGE_HEIGHT_PORTRAIT (page 357)

XS_PRINT_REPORT_PAGE_HEIGHT_PORTRAIT

Category

Templates and symbols

Use to specify the number of rows in printed reports with portrait orientation. The default value is `62`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_PRINT_REPORT_LINE_WIDTH_LANDSCAPE (page 356)

XS_PRINT_REPORT_LINE_WIDTH_PORTRAIT (page 356)

XS_PRINT_REPORT_PAGE_HEIGHT_LANDSCAPE (page 356)

XS_PRODUCT_IDENTIFIER

Category: Drawing View

To make it clear with which software a project has been modeled (old XSteel, other detailing systems or Tekla Structures) and reinforce the Tekla Structures brand, you can add a Tekla Structures product identifier to the side of every drawing. The product identifier will help promote the image of your company as a forward thinking company that uses the latest and most advanced technologies and techniques.

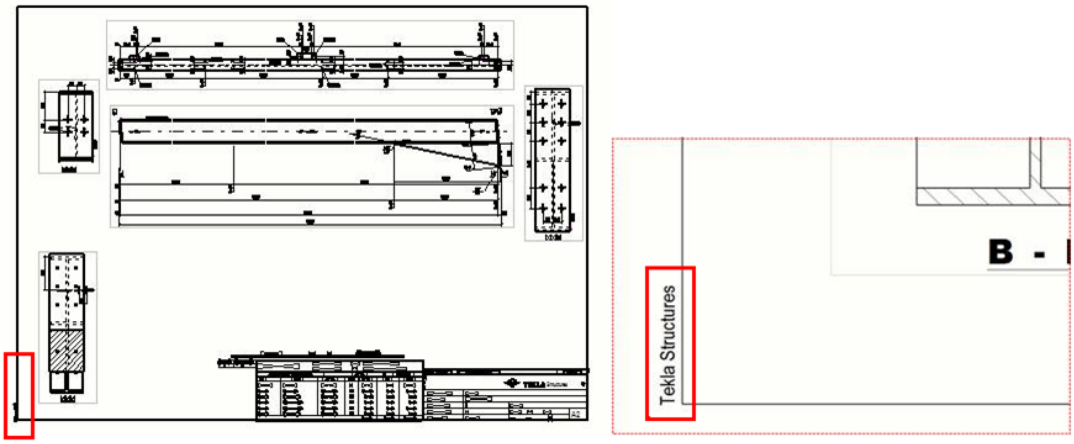
You can use the following values to change the position of the product identifier or turn it off: DX and DY offset, `FALSE`, and `TRUE` (default).

- If you do not want to use the product identifier, set this advanced option to `FALSE`.

- If you want to move the identifier, enter millimeter values separated by a comma (,) for both the X and Y directions.

  For example, `-5,10` will move the text 5 millimeters to the left and 10 millimeters up.

This advanced option is model specific and the setting is saved in the options database.



XS_PROFDB

Category

File Locations

Use to point to the profile folder which Tekla Structures searches for profile, material, device, and bolt catalogs.

You can store several catalogs in different locations, so it is important to know which catalog you are using.

This is a system-specific advanced option.

See also

XS_PROFILE_ANALYSIS_CHECK_ALL

Category: Analysis & Design

In the **Modify Profile Catalog** dialog box, you can enter analysis values for each profile. When you run the structural analysis, the analysis applications that use the COM link calculate the analysis values and compare them to the values in the Tekla Structures profile catalog. If the analysis application finds the values in the profile catalog, it uses the catalog values.

To check the profile catalog for analysis values for all profiles, enter `TRUE` for the following advanced options before you run the analysis:

- `XS_PROFILE_ANALYSIS_CHECK_ALL`
- `XS_AD_OPTIMISATION_DISABLED`

The default value is `FALSE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

If the profile catalog value differs significantly from the value the analysis application calculates, Tekla Structures writes a warning in the analysis log file. Use the advanced option `XS_PROFILE_ANALYSIS_VALUE_DIFF_LIMIT` to define the warning limit.

See also

XS_PROFILE_ANALYSIS_VALUE_DIFF_LIMIT (page 359)

XS_AD_OPTIMISATION_DISABLED (page 67)

XS_PROFILE_ANALYSIS_VALUE_DIFF_LIMIT

Category

Analysis & Design

Sets the warning limit as a percentage when checking profile catalog analysis values. The default value is `5.5` (%).

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_PROFILE_ANALYSIS_CHECK_ALL (page 358)

XS_PROFILE_DISPLAY_INCH_MARK_AFTER_FRACTIONS_IN_REPORTS

Category

Imperial Units

Use to define the location of the inch mark in profile lengths in reports.

To display the inch mark after the fractions (for example, PL1"X18 1/2"), enter `TRUE`. To display the inch mark before the fractions (for example, PL1"X18"1/2), enter `FALSE`.

The inch mark is displayed after the fractions by default (`TRUE`).

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XSR_SHOW_INCH_MARK_IN_PROFILE_NAMES (page 417)

XS_PROJECT

Category

File Locations

NOTE This advanced option is only meant for administrators.

Set the advanced options `XS_PROJECT` and `XS_FIRM`, along with `XS_SYSTEM`, to point to the folders Tekla Structures searches for properties files. Tekla Structures always saves properties in the current `model\attributes` folder. You can then copy or move them to the `XS_FIRM` or the `XS_PROJECT` folders if the same settings are needed in other models. You can also create user-defined sub-folders under the `XS_FIRM` and the `XS_PROJECT` folders, and copy or move property files from the `model\attributes` folder to these sub-folders.

For shared models, you can use a sub-folder in a Trimble Connect project as the project folder. For more information, see -

This advanced option is model specific and the setting is saved in the options database.

WARNING Changing an advanced option value in `.ini` files located outside the model folder does not affect the existing models. You can only update advanced options in the **Advanced Options** dialog box or in the `options.ini` file located in model folder; not from an `options.ini` file located in folders defined for the advanced options `XS_FIRM` or `XS_PROJECT`. The `.ini` files are read also when you open an existing model, but only new advanced options that do not exist in `options_model.db` or `options_drawings.db` are inserted, for example, such options that are not yet in the **Advanced Options** dialog box but have been added in the software.

See also

XS_PROTECT_SYMBOLS

Category

Drawing Properties

Set this advanced option to `TRUE` to prevent Tekla Structures from drawing objects on top of symbols. If you set it to `FALSE`, symbols are not protected. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

1.17 Advanced options - R

XS_RADIUS_TEXT_IN_UNFOLDING_BENDING_LINE_DIMENSIONING

Category: Dimensioning: unfolding

Use to set prefix text for radii. Enter any text, for example, `R=`. By default there is no prefix for radius in bending line dimensioning

This advanced option is model specific and the setting is saved in the options database.

XSR_BOLT_LENGTH_USE_ONLY_INCHES

Category

Templates and symbols

Set this advanced option to `TRUE` to prevent the advanced option `XSR_USE_ZERO_FEET_VALUE` from affecting bolt length in bolt marks. If you want `XSR_USE_ZERO_FEET_VALUE` to affect bolt length in bolt marks, set it to `FALSE`. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

See also

XSR_USE_ZERO_FEET_VALUE (page 396)

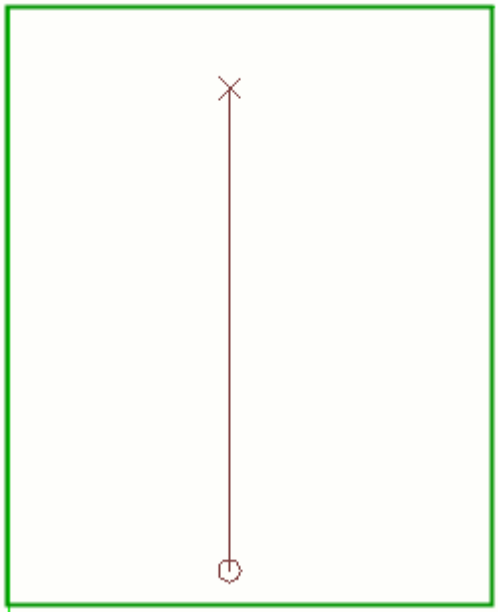
XS_REBAR_BEND_MARK_SYMBOL_MIN_SIZE

Category

Concrete Detailing

Use to increase the size of the reinforcing bar bend symbols in drawings (in drawing units) in order to see them more clearly. The default value is `1`.

The value set for this advanced option is multiplied by the view scale. If the resulting value is bigger than the default size (diameter of the reinforcing bar), then it will be used as the symbol size. Otherwise the default value is used. This means that to make the symbol as small as possible, leave the value out or use zero (0).



This advanced option can be used together with the advanced option `XS_REBAR_END_SYMBOL_MIN_SIZE`, which is used for increasing the size of the reinforcing bar end symbols.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_REBAR_END_SYMBOL_MIN_SIZE (page 364)

XS_REBAR_COMBINE_BENDINGS_IN_EVALUATOR

Category: **Concrete Detailing**

Use this advanced option to control how multiple sequential bendings in reinforcing bars are handled in **Rebar shape manager**.

When this advanced option is set to `TRUE` (which is the default value), multiple sequential bendings that form an arc are combined into one or more bendings (90 degrees or less) with the arc radius. This makes it possible to define bar bending shapes containing a large radius bending no matter how many individual bendings appear in the original bar geometry.

When the advanced option is set to `TRUE`, you can use the **Curve tolerance** setting in **Rebar shape manager** to define whether bendings are combined.

If you set the advanced option to `FALSE`, or if **Curve tolerance** is set to 0, bendings are not combined, but they appear as multiple bendings.

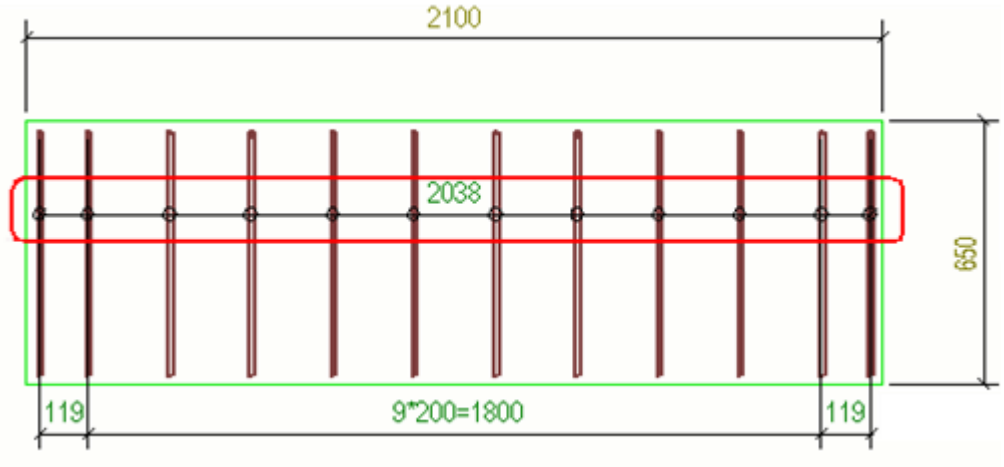
This advanced option is model specific and the setting is saved in the options database.

XS_REBAR_DIMENSION_LINE_SYMBOL

Category

This advanced option must be set in an initialization (.ini) file. Add this advanced option in the `options.ini` file under model folder.

Use this advanced option to change the reinforcing bar group dimension (distribution) line symbol. You can create the dimension line by right-clicking the reinforcing bar group and selecting **Create dimension line**. The default value is `xsteel@16`, where`xsteel` is the name of the symbol file and `16` is the running number of the symbol.



See also

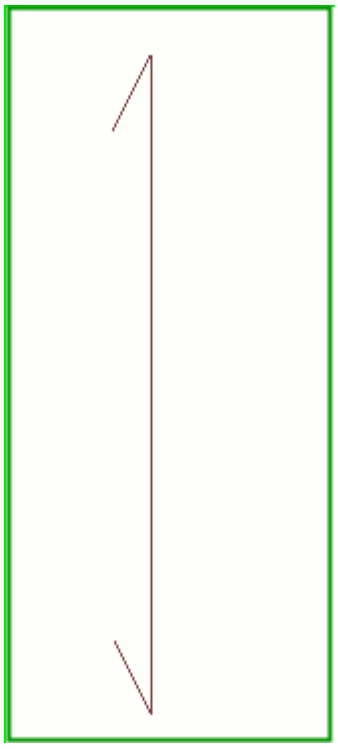
XS_REBAR_END_SYMBOL_MIN_SIZE

Category

Concrete Detailing

Use to increase the size of the reinforcing bar end symbols in drawings (in drawing units) in order to see them more clearly. Works for 45 or 135 degree symbols. The default value is `2`.

The value set for this advanced option is multiplied by the view scale. If the resulting value is bigger than the default size (diameter of the reinforcing bar), then it will be used as the symbol size. Otherwise the default value is used. This means that to make the symbol as small as possible, leave the value out or use zero (0).



This advanced option can be used together with the advanced option `XS_REBAR_BEND_MARK_SYMBOL_MIN_SIZE`, which is used for increasing the size of the reinforcing bar bend symbols.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_REBAR_REVERSE_END_SYMBOLS (page 367)

XS_REBAR_BEND_MARK_SYMBOL_MIN_SIZE (page 362)

XS_REBAR_MARK_LEADER_LINE_BASE_POINT_SEARCH_STEP_LENGTH

Category

Concrete Detailing

Use to define the step length while searching for an optimal place for the rebar mark leader line base point along the reinforcing bar. Enter the value in millimeters using decimals. The default value is `20.0`.

This advanced option is model specific and the setting is saved in the options database.

XS_REBAR_MARK_LEADER_LINE_BASE_POINT_SEARCH_TOLERANCE

Category

Concrete Detailing

Use to define how far the other reinforcing bars must be from the base point so that Tekla Structures can place the base point. Enter the value in millimeters using decimals. The default value is `10.0`.

This advanced option is model specific and the setting is saved in the options database.

XS_REBAR_POSITION_NUMBER_FORMAT_STRING

Category: Numbering

Use to define the contents of the reinforcing bar position numbers. You can also change or remove the separator and affect how many numbers are used to represent the reinforcing bar position number. If you change the value, you need to renumber the model.

Use the following options or a combination of them:

Option	Description
`%PART_PREFIX%`	The prefix of the position number of the part that contains the reinforcing bar.

Option	Description
%PART_START_NUMBER%	The start number of the position number of the part that contains the reinforcing bar.
%REBAR_PREFIX%	The prefix of the position number of the reinforcing bar.
%REBAR_SERIAL_NUMBER%	The position number without the prefix of the reinforcing bar.
%REBAR_POS%	This is no longer used. Use %REBAR_SERIAL_NUMBER% instead.
%REBAR_SIZE%	The size of the reinforcing bar with the possible size prefix. For example, in the US environments the size prefix is #.
%REBAR_SIZE_NUMBER%	The size of the reinforcing bar without the size prefix.
%CAST_UNIT_PREFIX%	The prefix of the position number of the cast unit that contains the reinforcing bar.
%CAST_UNIT_START_NUMBER%	The start number of the position number of the cast unit that contains the reinforcing bar.

This advanced option is model specific and the setting is saved in the options database.

Example

In the reinforcing bar properties, the **Prefix** is set to R, the **Start number** is set to 1, and the **Size** is set to #6.

- If you set the advanced option to %REBAR_SIZE%%REBAR_PREFIX% %REBAR_SERIAL_NUMBER.3%, the result for the first reinforcing bar will be #6R001.
- If you set the advanced option to %REBAR_SIZE_NUMBER%%REBAR_PREFIX %%REBAR_SERIAL_NUMBER.3% and number the model, the result for the first reinforcing bar is 6R001.

XS_REBAR_PULLOUT_ANGLE_TEXT_FRAME

Category

Concrete Detailing

Use to turn off the text frame around the angle text in pull-out pictures. By default, this advanced option is set to FALSE, and no frame is drawn. If you set this advanced option to TRUE, the frame is drawn.

If you have set XS_REBAR_PULLOUT_ANGLE_TEXT_UNDERLINE to TRUE, then the advanced option XS_REBAR_PULLOUT_ANGLE_TEXT_FRAME is ignored.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_REBAR_PULLOUT_ANGLE_TEXT_UNDERLINE (page 367)

XS_REBAR_PULLOUT_ANGLE_TEXT_UNDERLINE

Category

Concrete Detailing

Set this advanced option to TRUE to draw an line under the angle text in pull-outs. If you set it to TRUE, then the advanced option XS_REBAR_PULLOUT_ANGLE_TEXT_FRAME is ignored. By default, this advanced option is set to FALSE, and no underline is drawn.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_REBAR_PULLOUT_ANGLE_TEXT_FRAME (page 366)

XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION

Category

Concrete Detailing

Set to FALSE to have Tekla Structures ignore hooks when it checks the shape of reinforcing bars, and to have Tekla Structures assign the same bending type to bars with and without hooks.

Set to TRUE to have Tekla Structures take the hooks into account and to treat bars with or without hooks, or with different hooks, as different.

The default value is TRUE.

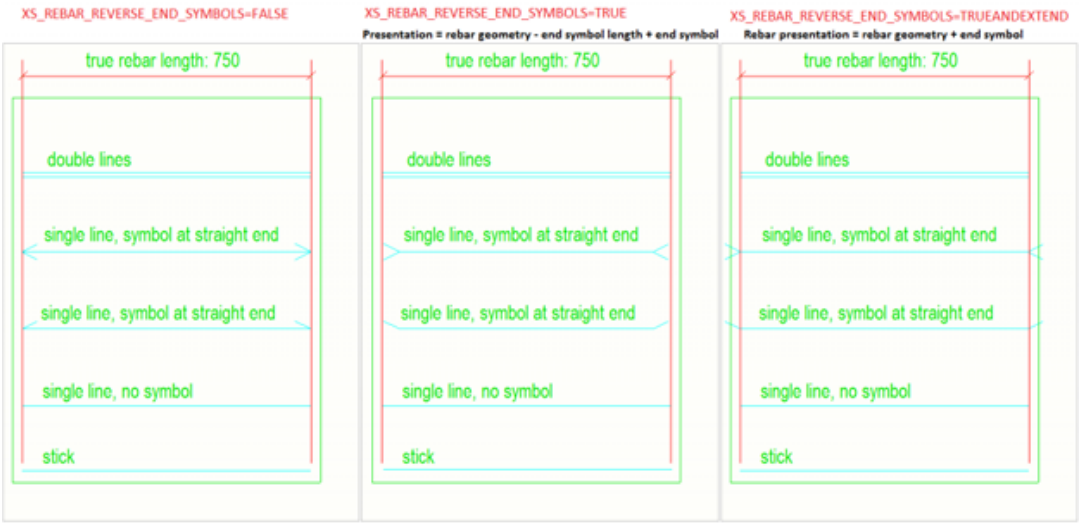
This advanced option is model specific and the setting is saved in the options database.

See also

XS_REBAR_REVERSE_END_SYMBOLS

Category: Concrete Detailing

Use to reverse the reinforcing bar end symbols to a different direction. When this advanced option is set to `TRUE`, the end symbol is drawn at 135 degree angle (used commonly in Norway). If you use single line visualization and no symbol at straight end, use the value `TRUEANDEXTEND`. If you use `TRUE` for these kinds of reinforcing bars, they will be drawn too short. The default value is `FALSE`.



This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

See also

XS_REBAR_END_SYMBOL_MIN_SIZE (page 364)

Reinforcement object properties in drawings (page 768)

XS_REBARSET_BUFFER_SIZE

Category: Speed and Accuracy

This advanced option defines the size of the memory cache used to store rebar set bars. When you increase the size, more rebar set bars can be kept in memory. This means that the rebar set bars are not regenerated so frequently, which will improve the performance. For optimum performance,

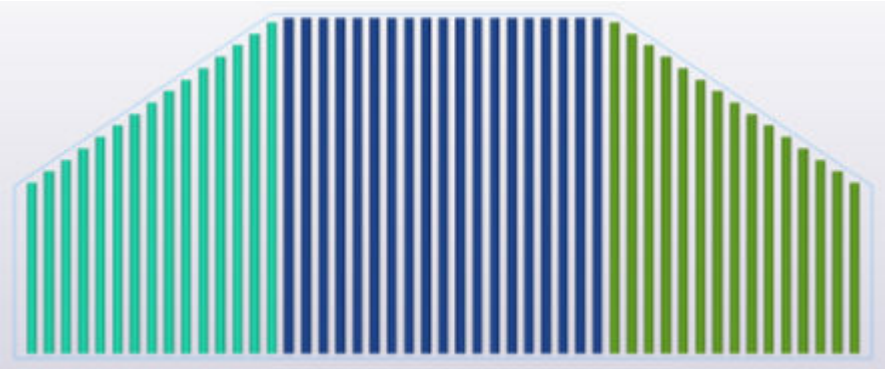
the size should be greater or equal to the number of rebar set bars in the model.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_REBARSET_COLOR_BARGROUPS

Category: Concrete Detailing

Set this advanced option to `TRUE` to show bar groups within a rebar set using different colors in model views. For example:



When you set this advanced option to `TRUE`, the reinforcing bars in rebar sets will not be colored by class.

The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

After changing the setting of this advanced option in the **Advanced Options** dialog box, redraw the model views.

TIP To quickly switch between the values `TRUE` and `FALSE`, go to the **Rebar** tab on the ribbon and click **Visibility** --> **Color groups**, or use the keyboard shortcut **Alt+7**.

XS_REBARSET_CREATION_ANGLE_TOLERANCE_FOR_CROSSING_REBARS

Category: Concrete Detailing

Use this advanced option to define the tolerance for the angle between consecutive part faces when the rebar set bars are created. If the angle between a part face and the extension of the previous part face is less than the value of this advanced option, a rebar set leg face is created at the part face.

This advanced option applies to the rebar sets when they are created using the **Create crossing rebars** command. The default value is 45 (degrees).

This advanced option is model specific and the setting is saved in the options database.

See also

XS_REBARSET_CREATION_ANGLE_TOLERANCE_FOR_LONGITUDINAL_REBARS (page 370)

XS_REBARSET_CREATION_ANGLE_TOLERANCE_FOR_LONGITUDINAL_REBARS

Category: Concrete Detailing

Use this advanced option to define the tolerance for the angle between consecutive part faces when the rebar set bars are created. If the angle between a part face and the extension of the previous part face is less than the value of this advanced option, a rebar set leg face is created at the part face.

This advanced option applies to the rebar sets when they are created using the **Create longitudinal rebars** command. The default value is 45 (degrees).

This advanced option is model specific and the setting is saved in the options database.

See also

XS_REBARSET_CREATION_ANGLE_TOLERANCE_FOR_CROSSING_REBARS (page 369)

XS_REBARSET_ENABLE_BAR_GROUPING_WHEN_SPACING_DIFFERS

Category: Concrete Detailing

Use this advanced option to define whether spacing affects how reinforcing bars are grouped in rebar sets.

The default value is `TRUE`, which means that similar bars in adjacent spacing zones in a rebar set are grouped even when the spacing differs.

If this advanced option is set to `FALSE`, each spacing zone in a rebar set automatically creates a separate group.

This advanced option is model specific and the setting is saved in the options database.

After changing the setting of this advanced option, you need to update the existing rebar sets in the model. On the **Rebar** tab on the ribbon, click **More** --> **Regenerate** to activate the new setting.

XS_REBARSET_LEG_CONNECTION_TOLERANCE

Category: Concrete Detailing

Use this advanced option to connect rebar set bar legs together even though the leg face edges do not overlap exactly. Define the maximum gap between the leg faces that are automatically connected. If the value is bigger than the existing gap, the gap is ignored and the bar legs are connected.

The default value is 1.0 mm.

This advanced option is model specific and the setting is saved in the options database.

After changing the value of this advanced option, you need to update the existing rebar sets in the model. On the **Rebar** tab on the ribbon, click **More** --> **Regenerate** to activate the new value.

XS_REBARSET_MINIMUM_LEG_DEVIATION

Category: Concrete Detailing

Use this advanced option to define whether rounding settings are applied to certain reinforcing bar legs in rebar sets or not.

Tekla Structures compares each bar leg to a straight line. If the deviation of a leg from the line is less than the value of this advanced option, the leg is

considered to be a part of a curved bar segment and the leg length is not rounded.

If the deviation is more than the value of this advanced option, Tekla Structures considers the leg to be a straight bar segment and then rounds the leg length according to the rounding settings.

Enter the value in millimeters. The default value is `10`.

This advanced option is model specific and the setting is saved in the options database.

XS_REBARSET_REBAR_LAYER_FORMAT_STRING

Category: Concrete Detailing

Use this advanced option to define the contents of the template attribute LAYER (page 565).

The default value is `%LAYER_PREFIX%%LAYER_NUMBER%`, for example T2 for the second layer of top bars.

This advanced option is model specific and the setting is saved in the options database.

See also

LAYER_PREFIX (page 566)

LAYER_NUMBER (page 565)

XS_REBARSET_SHOW_END_DETAIL_MODIFIERS

Category: Concrete Detailing

Set this advanced option to `TRUE` to show the rebar set end detail modifiers when you select rebar set bars in the model.

If this advanced option is set to `FALSE`, end detail modifiers are not shown when you select rebar set bars.

The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

TIP To quickly switch between the values `TRUE` and `FALSE`, go to the **Rebar** tab on the ribbon and click **Visibility** --> **End detail modifiers**, or use the keyboard shortcut **Alt+5**.

See also

XS_REBARSET_SHOW_PROPERTY_MODIFIERS (page 374)

XS_REBARSET_SHOW_SPLITTERS (page 375)

XS_REBARSET_SHOW_GUIDELINES (page 373)

XS_REBARSET_SHOW_MODIFIERS_CREATED_BY_COMPONENTS (page 374)

XS_REBARSET_SHOW_GUIDELINES

Category: Concrete Detailing

Set this advanced option to `TRUE` to show the rebar set guidelines when you select rebar set bars in the model.

If this advanced option is set to `FALSE`, rebar set guidelines are not shown when you select rebar set bars.

The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

TIP To quickly switch between the values `TRUE` and `FALSE`, go to the **Rebar** tab on the ribbon and click **Visibility** --> **Guidelines**, or use the keyboard shortcut **Alt+2**.

See also

XS_REBARSET_SHOW_PROPERTY_MODIFIERS (page 374)

XS_REBARSET_SHOW_END_DETAIL_MODIFIERS (page 372)

XS_REBARSET_SHOW_SPLITTERS (page 375)

XS_REBARSET_SHOW_MODIFIERS_CREATED_BY_COMPONENTS (page 374)

XS_REBARSET_SHOW_LEGFACES

Category: Concrete Detailing

Set this advanced option to `TRUE` to show the rebar set leg faces and leg surfaces when you select rebar set bars in the model.

If this advanced option is set to `FALSE`, leg faces and leg surfaces are not shown when you select rebar set bars.

The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

TIP To quickly switch between the values TRUE and FALSE, go to the **Rebar** tab on the ribbon and click **Visibility** --> **Leg faces**, or use the keyboard shortcut **Alt+1**.

XS_REBARSET_SHOW_MODIFIERS_CREATED_BY_COMPONENTS

Category: Concrete Detailing

Use this advanced option to control whether the rebar set modifiers that are created by components are shown or hidden in model views when you select rebar set bars.

The default value is FALSE, which means that the modifiers are hidden.

Note that if you explode a component that has modifiers, the modifiers will be shown even if this advanced option is set to FALSE.

This advanced option does not affect the custom component views.

This advanced option is model specific and the setting is saved in the options database.

XS_REBARSET_SHOW_PROPERTY_MODIFIERS

Category: Concrete Detailing

Set this advanced option to TRUE to show the rebar set property modifiers when you select rebar set bars in the model.

If this advanced option is set to FALSE, property modifiers are not shown when you select rebar set bars.

The default value is TRUE.

This advanced option is model specific and the setting is saved in the options database.

TIP To quickly switch between the values TRUE and FALSE, go to the **Rebar** tab on the ribbon and click **Visibility** --> **Property modifiers**, or use the keyboard shortcut **Alt+3**.

See also

XS_REBARSET_SHOW_END_DETAIL_MODIFIERS (page 372)

XS_REBARSET_SHOW_SPLITTERS (page 375)
XS_REBARSET_SHOW_GUIDELINES (page 373)
XS_REBARSET_SHOW_MODIFIERS_CREATED_BY_COMPONENTS (page 374)

XS_REBARSET_SHOW_SPLITTERS

Category: Concrete Detailing

Set this advanced option to `TRUE` to show the rebar set splitters when you select rebar set bars in the model.

If this advanced option is set to `FALSE`, splitters are not shown when you select rebar set bars.

The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

TIP To quickly switch between the values `TRUE` and `FALSE`, go to the **Rebar** tab on the ribbon and click **Visibility** --> **Splitters**, or use the keyboard shortcut **Alt+4**.

See also

XS_REBARSET_SHOW_PROPERTY_MODIFIERS (page 374)
XS_REBARSET_SHOW_END_DETAIL_MODIFIERS (page 372)
XS_REBARSET_SHOW_GUIDELINES (page 373)
XS_REBARSET_SHOW_MODIFIERS_CREATED_BY_COMPONENTS (page 374)

XS_REBARSET_SIMILAR_GROUPING_NUMBER

Category: Concrete Detailing

Use this advanced option to define the minimum number of similar rebar set bars that can be automatically grouped.

The default value is 3.

This advanced option is model specific and the setting is saved in the options database.

After changing the value of this advanced option, you need to update the existing rebar sets in the model. On the **Rebar** tab on the ribbon, click **More** --> **Regenerate** to activate the new value.

If you need to override the value of this advanced option in certain rebar sets, enter a value for **Minimum number of bars in similar group** in the rebar set or property modifier user-defined attributes.

See also

XS_REBARSET_SIMILAR_GROUPING_TOLERANCE (page 376)

XS_REBARSET_SIMILAR_GROUPING_TOLERANCE

Category: Concrete Detailing

Use this advanced option to define which similar rebar set bars can be automatically grouped. Define the maximum allowed difference in the geometry of the bars that will be grouped.

The default value is 1.0 mm.

This advanced option is model specific and the setting is saved in the options database.

After changing the value of this advanced option, you need to update the existing rebar sets in the model. On the **Rebar** tab on the ribbon, click **More** --> **Regenerate** to activate the new value.

See also

XS_REBARSET_TAPERED_LINEAR_GROUPING_TOLERANCE (page 378)

XS_REBARSET_TAPERED_CURVED_GROUPING_TOLERANCE (page 376)

XS_REBARSET_SIMILAR_GROUPING_NUMBER (page 375)

XS_REBARSET_TAPERED_CURVED_GROUPING_TOLERANCE

Category: Concrete Detailing

Use this advanced option to define which rebar set bars can automatically form a tapered bar group that follows a curve. Either one end or both ends of the grouped bars can follow a curve. Define the maximum allowed distance of the bar ends from the curve.

The default value is 10 mm.

This advanced option is model specific and the setting is saved in the options database.

After changing the value of this advanced option, you need to update the existing rebar sets in the model. On the **Rebar** tab on the ribbon, click **More** --> **Regenerate** to activate the new value.

If you need to override the value of this advanced option in certain rebar sets, enter a value for **Tapered curved tolerance** in the rebar set or property modifier user-defined attributes.

See also

XS_REBARSET_TAPERED_LINEAR_GROUPING_TOLERANCE (page 378)

XS_REBARSET_SIMILAR_GROUPING_TOLERANCE (page 376)

XS_REBARSET_TAPERED_GROUP_POSITION_NUMBER_FORMAT_STRING

Category: Numbering

Use this advanced option to define the contents of the reinforcement position numbers (GROUP_POS (page 553)) in tapered bar groups within rebar sets. You can also change or remove the separator and affect how many numbers are used to represent the position number. If you change the value, you need to renumber the model.

Use the following options or a combination of them:

Option	Description
%PART_PREFIX%	The prefix of the position number of the part that contains the reinforcing bar.
%PART_START_NUMBER%	The start number of the position number of the part that contains the reinforcing bar.
%REBAR_PREFIX%	The prefix of the position number of the reinforcing bar.
%REBAR_SERIAL_NUMBER%	The position number without the prefix of the reinforcing bar.
%REBAR_POS%	This is no longer used. Use %REBAR_SERIAL_NUMBER% instead.
%REBAR_SIZE%	The size of the reinforcing bar with the possible size prefix. For example, in the US environments the size prefix is #.
%REBAR_SIZE_NUMBER%	The size of the reinforcing bar without the size prefix.
%CAST_UNIT_PREFIX%	The prefix of the position number of the cast unit that contains the reinforcing bar.

Option	Description
%CAST_UNIT_START_NUMBER%	The start number of the position number of the cast unit that contains the reinforcing bar.

The default value is %REBAR_PREFIX%%REBAR_SERIAL_NUMBER%.

This advanced option is model specific and the setting is saved in the options database.

Example

The **Prefix** in the rebar set properties is set to R, the **Start number** is set to 1, and the **Size** is set to #6.

- If you set the advanced option to %REBAR_SIZE%%REBAR_PREFIX% %REBAR_SERIAL_NUMBER.3%, the result for the first reinforcing bar will be #6R001.
- If you set the advanced option to %REBAR_SIZE_NUMBER%%REBAR_PREFIX %%REBAR_SERIAL_NUMBER.3% and number the model, the result for the first reinforcing bar is 6R001.

See also

XS_REBARSET_TAPERED_REBAR_POSITION_NUMBER_FORMAT_STRING (page 379)

XS_REBARSET_TAPERED_LINEAR_GROUPING_TOLERANCE

Category: Concrete Detailing

Use this advanced option to define which rebar set bars can automatically form a tapered bar group that follows a line. Define the maximum allowed distance of the bar ends from the line.

The default value is 0.5 mm.

This advanced option is model specific and the setting is saved in the options database.

After changing the value of this advanced option, you need to update the existing rebar sets in the model. On the **Rebar** tab on the ribbon, click **More** --> **Regenerate** to activate the new value.

If you need to override the value of this advanced option in certain rebar sets, enter a value for **Tapered linear tolerance** in the rebar set or property modifier user-defined attributes.

See also

XS_REBARSET_TAPERED_CURVED_GROUPING_TOLERANCE (page 376)

XS_REBARSET_SIMILAR_GROUPING_TOLERANCE (page 376)

XS_REBARSET_TAPERED_REBAR_POSITION_NUMBER_FORMAT_STRING

Category: Numbering

Use this advanced option to define the contents of the reinforcement position numbers (REBAR_POS (page 586)) of single bars in tapered bar groups within rebar sets. You can also change or remove the separator and affect how many numbers are used to represent the position number. If you change the value, you need to renumber the model.

Use the following options or a combination of them:

Option	Description
%PART_PREFIX%	The prefix of the position number of the part that contains the reinforcing bar.
%PART_START_NUMBER%	The start number of the position number of the part that contains the reinforcing bar.
%REBAR_PREFIX%	The prefix of the position number of the reinforcing bar.
%REBAR_SERIAL_NUMBER%	The position number without the prefix of the reinforcing bar.
%REBAR_POS%	This is no longer used. Use %REBAR_SERIAL_NUMBER% instead.
%REBAR_SIZE%	The size of the reinforcing bar with the possible size prefix. For example, in the US environments the size prefix is #.
%REBAR_SIZE_NUMBER%	The size of the reinforcing bar without the size prefix.
%CAST_UNIT_PREFIX%	The prefix of the position number of the cast unit that contains the reinforcing bar.
%CAST_UNIT_START_NUMBER%	The start number of the position number of the cast unit that contains the reinforcing bar.
%SUB_ID%	The running index number of the reinforcing bar in a tapered bar group within a rebar set.
%SUB_ID_WITH_LETTERS%	Same as above, but with letters. Uses letters A–Z by default, but you can also define the valid letters with the advanced option XS_VALID_CHARS_FOR_REBAR_SUB_ID_WITH_LETTERS (page 493).

The default value is `%REBAR_PREFIX%%REBAR_SERIAL_NUMBER%.%SUB_ID%`.

This advanced option is model specific and the setting is saved in the options database.

Example

The **Prefix** in the rebar set properties is set to `R`, the **Start number** is set to `1`, and the **Size** is set to `#6`.

- If you set the advanced option to `%REBAR_SIZE%%REBAR_PREFIX% %REBAR_SERIAL_NUMBER.3%`, the result for the first reinforcing bar will be `#6R001`.
- If you set the advanced option to `%REBAR_SIZE_NUMBER%%REBAR_PREFIX %%REBAR_SERIAL_NUMBER.3%` and number the model, the result for the first reinforcing bar is `6R001`.

See also

XS_REBARSET_TAPERED_GROUP_POSITION_NUMBER_FORMAT_STRING (page 377)

XS_REBARSET_USE_GROUP_NUMBER_FOR_BARS_IN_TAPERED_GROUPS

Category: Numbering

Use this advanced option to define whether reinforcing bars in tapered bar groups within rebar sets are numbered using their group numbers, or as individual bars.

The default value is `TRUE`, which means that each bar in a tapered bar group is numbered using the group number.

If this advanced option is set to `FALSE`, bars in tapered bar groups are numbered as individual bars.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_REBARSET_TAPERED_GROUP_POSITION_NUMBER_FORMAT_STRING (page 377)

XS_REBARSET_TAPERED_REBAR_POSITION_NUMBER_FORMAT_STRING (page 379)

XS_REBAR_USE_ALWAYS_METHOD_A_FOR_90_DEGREE_HOOK_DIMENSIONS

This advanced option must be set in an initialization (.ini) file.

When set to `TRUE`, the **SHLB/EHLB** hook properties will have the same values as the **SHLA/EHLA** properties in case the hook angle is approximately 95 degrees or less.

XS_REBAR_USE_POLYCURVES_IN_EVALUATOR

Category: Concrete Detailing

When this advanced option is set to `TRUE`, which is the default value, **Rebar shape manager** recognizes and uses the polycurve geometries of reinforcing bars. Polycurves can consist of straight and curved segments.

If this advanced option is set to `FALSE`, polycurve bar geometries are recognized as polylines and may result in polygonal chains of short bar legs.

This advanced option is model specific and the setting is saved in the options database.

XS_RECREATE_MARKS_IN_INTELLIGENT_CLONING

Category

Marking: General

Set this advanced option to `ALL` to recreate all marks during intelligent cloning. If you leave the value out, marks are not recreated. By default, no value is set.

This advanced option is model specific and the setting is saved in the options database.

XS_RECREATE_UNMODIFIED_DRAWINGS

Category: **Drawing Properties**

Use to define whether drawings are recreated when you update an assembly, single or cast unit drawing that has not been modified. Drawings are automatically recreated unless they have been edited and then saved, or they have been issued using the **Issue** functionality in **Document manager**.

- To prevent the recreation of the unmodified drawings, set the advanced option `FALSE`.

- To allow the recreation of the unmodified drawings, set the advanced option to `TRUE`. This is the default value.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_REFERENCE_CACHE

Category

File Locations

Use to define the default location of the cache file that is created from the source file when you load a reference model for the first time. The advanced option is set to `C:\TeklaStructuresModels\\RefCacheFolders` by default. You can also replace the path with `XS_RUNPATH` as follows: `%XS_RUNPATH%\RefCacheFolders`.

TIP
- You may sometimes want to change the default location of the cache file when you are working with multi-user models to reduce network traffic and disk usage in the server or to speed up the cache operation (if the local drive is faster than the server drive).
- When you are using different versions of Tekla Structures for different projects and you experience problems with reference models, empty the folder where the reference cache is created. The cache file is recreated the next time you open the reference model.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_REFERENCE_MODEL_KEEP_VERSIONS_COUNT

Category: Modeling Properties

Use the `XS_REFERENCE_MODEL_KEEP_VERSIONS_COUNT` advanced option to clean up old reference model revisions automatically. The cleanup is done when the reference file is updated. Use `XS_DELETE_UNNECESSARY_REFMODEL_FILES_SAFETY_PERIOD (page 149)` to set the time frame for deletion. Tekla Structures deletes the reference models that have been imported at some point but are no longer used, and are not displayed in the **Reference Models** list. The data related to these reference models is deleted from the current data storage in folder `<current`

model>\datastorage\ref. The original imported reference model is not deleted from its actual folder, for example, from .\Reference models.

You can use the following values:

- 0: The cleanup is disabled. This is the default value.
- Any positive number.

  For example, value 3 keeps two old revisions of the reference model, in addition to the current version.

This advanced option is model specific and the setting is saved in the options database.

XS_REFERENCE_USE_RENDERED_CLIPPING

Category

Import

Set to TRUE to have Tekla Structures show only the center line of reference objects outside the work area in model views. This can be useful, for example, when viewing cylindrical DGN structures, such as piping. If you do not want to show only center line, set it to FALSE. The default value is TRUE. I

Tekla Structures then shows the objects as follows:

- Objects entirely inside work area are rendered.
- Objects entirely outside work area are hidden.
- Objects partly inside the work area are rendered inside work area, and wireframe outside work area.

This advanced option is user specific and the setting is saved in options.bin under the user folder, for example, in C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings. Restart Tekla Structures to activate the new value.

XS_REFRESH_ALSO_LOCKED_REFERENCE_MODELS

Category: Import

Set the advanced option XS_REFRESH_ALSO_LOCKED_REFERENCE_MODELS to TRUE to refresh locked reference models with the **Refresh** button . This advanced option is by default set to FALSE.

This advanced option is system specific.

XS_REMEMBER_LAST_PLOT_DIALOG_VALUES

Category

This advanced option must be set in an initialization (.ini) file.

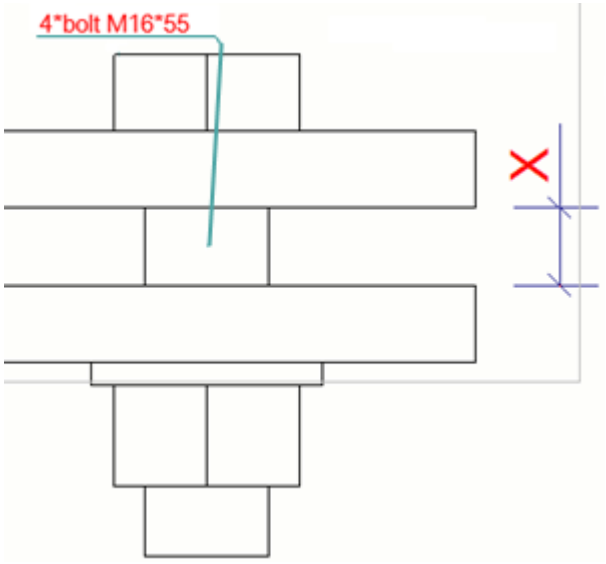
If you set this advanced option to `TRUE`, Tekla Structures remembers the latest settings that were used in the **Print Drawings** dialog box when you open the dialog box the next time. If you do not want to do this, enter `false`. The default value is `TRUE`.

XS_REMOVE_VOID_FROM_BOLT_MATERIAL_THICKNESS

Category: Modeling Properties

Set this advanced option to `TRUE` to remove the gap between two bolted material thicknesses, resulting in a shorter bolt length. This is required by, for example, mast builders. The default value is `FALSE`.

In the example below, the advanced option is set to `TRUE`. The value "X" will be removed from the bolt length.



This advanced option is model specific and the setting is saved in the options database.

XS_RENDERED_CURSOR_LINE_WIDTH

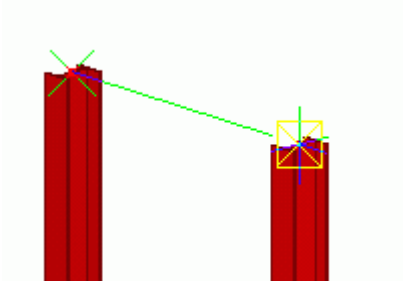
Category

Model View

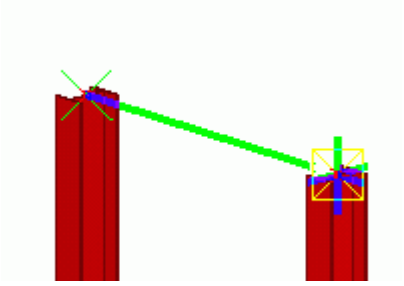
Use to set cursor line width in model views.

- Possible values are `1`, `2`, or `4`. Any other value is handled as 1.
- The default value is `2`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.



Cursor line width of **1**



Cursor line width of **4**

XS_RENDERED_FIELD_OF_VIEW

Category

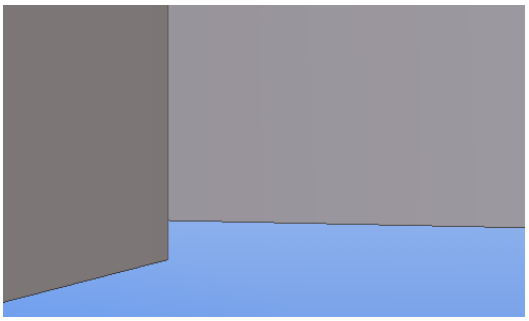
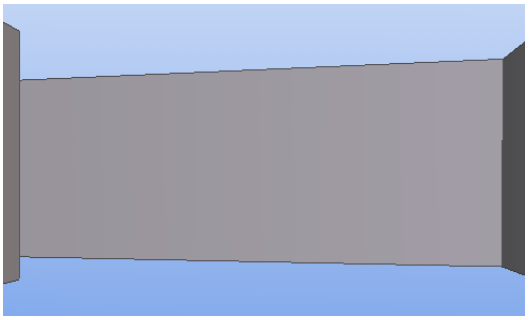
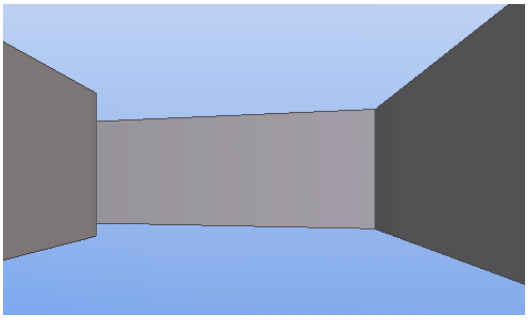
Model View

Use to adjust the field of view setting in perspective views. This can be useful, for example, when using the **Fly** command in a tight space. The bigger the value, the more distance there is between the parts.

The default value is `60.0`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

Example

Field of view	Example
60.0	
90.0	
120.0	

XS_RENDERED_GL_FOG_END_VALUE

Category

Model View

Note that this advanced option works only in views that use OpenGL rendering.

In model views, the distant objects appear progressively darker than the close ones. Use the advanced options `XS_RENDERED_GL_FOG_START_VALUE` and `XS_RENDERED_GL_FOG_END_VALUE` to control the shade of objects.

Use values from 0.0 to 1.0 to control the shade of objects. The higher the value, the darker the distant objects. Value 0 disables the fog effect. The default value for `XS_RENDERED_GL_FOG_END_VALUE` is `0`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_RENDERED_GL_FOG_START_VALUE (page 387)

XS_RENDERED_GL_FOG_START_VALUE

Category

Model View

Note that this advanced option works only if you use the default OpenGL rendering instead of the DirectX rendering.

In model views, the distant objects appear progressively darker than the close ones. Use the advanced options `XS_RENDERED_GL_FOG_START_VALUE` and `XS_RENDERED_GL_FOG_END_VALUE` to control the shade of objects.

Use values from 0.0 to 1.0 to control the shade of objects. The higher the value, the darker the distant objects. Value 0 disables the fog effect. The default value for `XS_RENDERED_GL_FOG_START_VALUE` is `0`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_RENDERED_GL_FOG_END_VALUE (page 386)

XS_RENDERED_PIXEL_TOLERANCE_SCALE

Category

Model View

Tekla Structures uses pixel tolerance to differentiate between clicking and dragging the mouse when zooming. Use this advanced option to define the pixel tolerance.

The default value is `0.7`. If the mouse moves less than the specified value while the left mouse button is pressed down, it is treated as a click.

This is a system-specific advanced option.

XS_REPORT_BOLTS_WITH_SUPPORTING_MEMBER

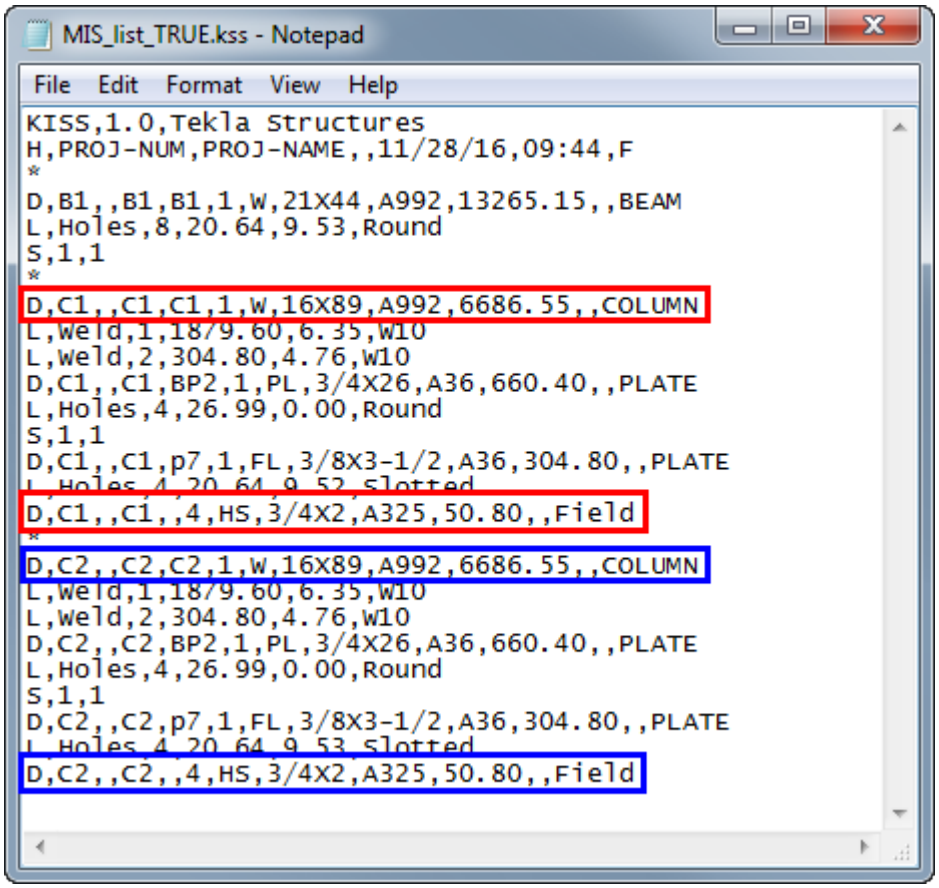
Category: Templates and symbols

You can set the site bolts to the supporting member in reports and KSS by setting the advanced option `XS_REPORT_BOLTS_WITH_SUPPORTING_MEMBER` to `TRUE`. Using this advanced option you can show the field bolts in the BOM of the supporting member. The default value is `FALSE`.

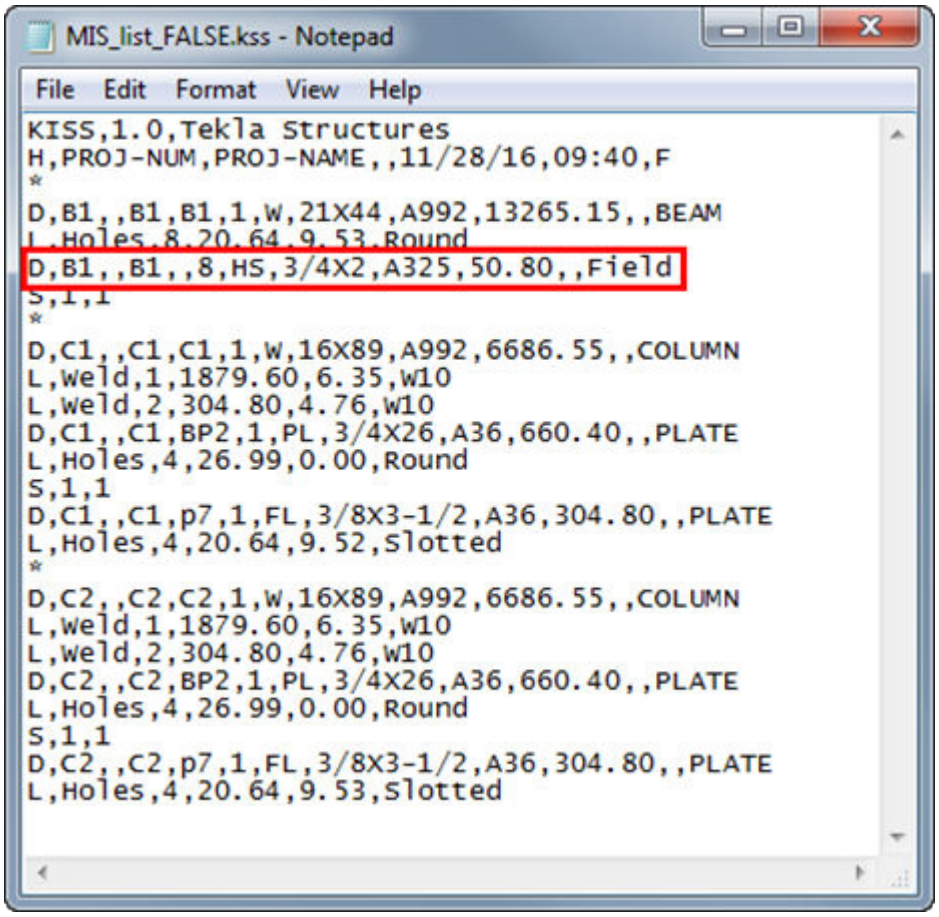
In the following BOM example, the advanced option is set to `TRUE`:

In the following BOM example, the advanced option is set to `FALSE`:

In the following KSS file example, the advanced option is set to `TRUE`:



In the following KSS file example, the advanced option is set to `FALSE`:



This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_REPORT_OUTPUT_DIRECTORY

Category

File locations

Points to the folder where Tekla Structures saves reports. If the full path appears in the report file name field, Tekla Structures ignores this setting. The default value is `.\Reports`.

XS_RESTORE_ENABLES

Category

This advanced option must be set in an initialization (.ini) file.

Set this advanced option to `TRUE` to be able to save and load check box values in dialog boxes. `FALSE` is the default value.

XS_RIBBON_CONFIGURATION_CARBON

This advanced option must be set in an initialization (.ini) file.

Use this advanced option to control which ribbon is shown for a specified role for those users who are using the Tekla Structures Carbon configuration. The role-specific initialization files (`role_<role>.ini`), for example, `role_Steel_Detailer.ini` or `role_Rebar_Detailer.ini` contain the advanced option.

The advanced option needs to point the configuration identifier of the ribbon file, either to the configuration identifier of the legacy on-premises license configuration, or to the configuration identifier of the online license configuration. It is not possible to make up new configuration identifiers.

Example:

For the users who select the **Precast Concrete Detailing** role, you could have the following settings in the `role_Precast_Detailer.ini` file:

```
set XS_RIBBON_CONFIGURATION_CARBON=albl_up_Carbon
set XS_RIBBON_CONFIGURATION_GRAPHITE=albl_up_PC_Detailing
set XS_RIBBON_CONFIGURATION_DIAMOND=albl_up_PC_Detailing
```

See also

XS_RIBBON_CONFIGURATION_GRAPHITE (page 392)

XS_RIBBON_CONFIGURATION_DIAMOND (page 391)

XS_RIBBON_CONFIGURATION_DIAMOND

This advanced option must be set in an initialization (.ini) file.

Use this advanced option to control which ribbon is shown for a specified role for those users who are using the Tekla Structures Diamond configuration. The role-specific initialization files (`role_<role>.ini`), for example,

role_Steel_Detailer.ini or role_Rebar_Detailer.ini contain the advanced option.

The advanced option needs to point the configuration identifier of the ribbon file, either to the configuration identifier of the legacy on-premises license configuration, or to the configuration identifier of the online license configuration. It is not possible to make up new configuration identifiers.

Example:

For the users who select the **Precast Concrete Detailing** role, you could have the following settings in the role_Precast_Detailer.ini file:

```
set XS_RIBBON_CONFIGURATION_CARBON=albl_up_Carbon
set XS_RIBBON_CONFIGURATION_GRAPHITE=albl_up_PC_Detailing
set XS_RIBBON_CONFIGURATION_DIAMOND=albl_up_PC_Detailing
```

See also

XS_RIBBON_CONFIGURATION_CARBON (page 391)

XS_RIBBON_CONFIGURATION_GRAPHITE (page 392)

XS_RIBBON_CONFIGURATION_GRAPHITE

This advanced option must be set in an initialization (.ini) file.

Use this advanced option to control which ribbon is shown for a specified role for those users who are using the Tekla Structures Graphite configuration. The role-specific initialization files (role_<role>.ini), for example, role_Steel_Detailer.ini or role_Rebar_Detailer.ini contain the advanced option.

The advanced option needs to point the configuration identifier of the ribbon file, either to the configuration identifier of the legacy on-premises license configuration, or to the configuration identifier of the online license configuration. It is not possible to make up new configuration identifiers.

Example:

For the users who select the **Precast Concrete Detailing** role, you could have the following settings in the role_Precast_Detailer.ini file:

```
set XS_RIBBON_CONFIGURATION_CARBON=albl_up_Carbon
set XS_RIBBON_CONFIGURATION_GRAPHITE=albl_up_PC_Detailing
set XS_RIBBON_CONFIGURATION_DIAMOND=albl_up_PC_Detailing
```

See also

XS_RIBBON_CONFIGURATION_CARBON (page 391)

XS_RIBBON_CONFIGURATION_DIAMOND (page 391)

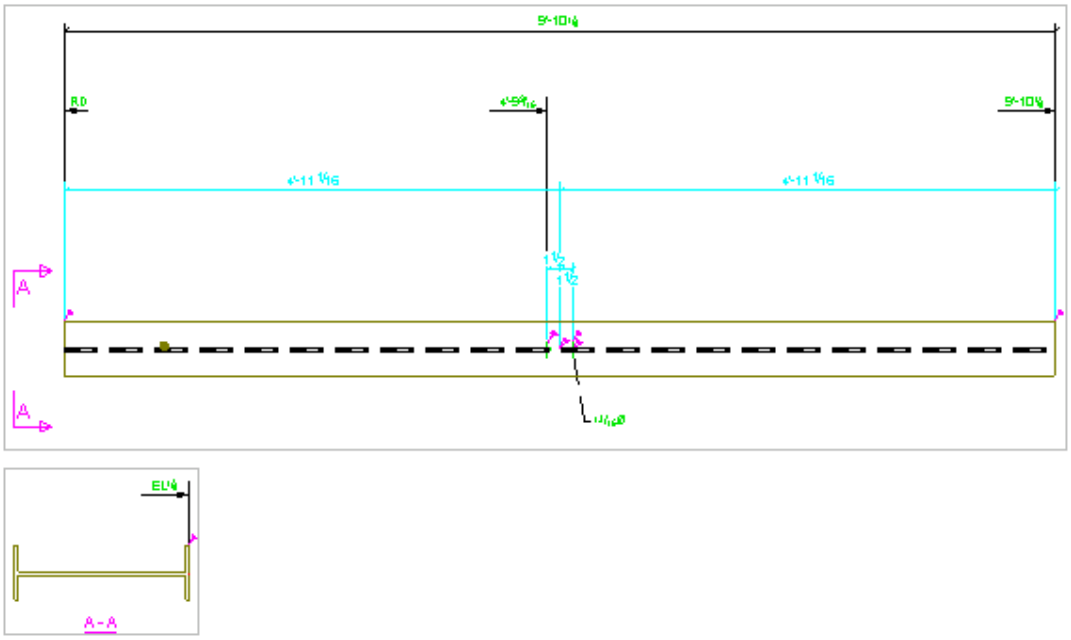
XS_ROTATE_CUT_VIEWS

Category

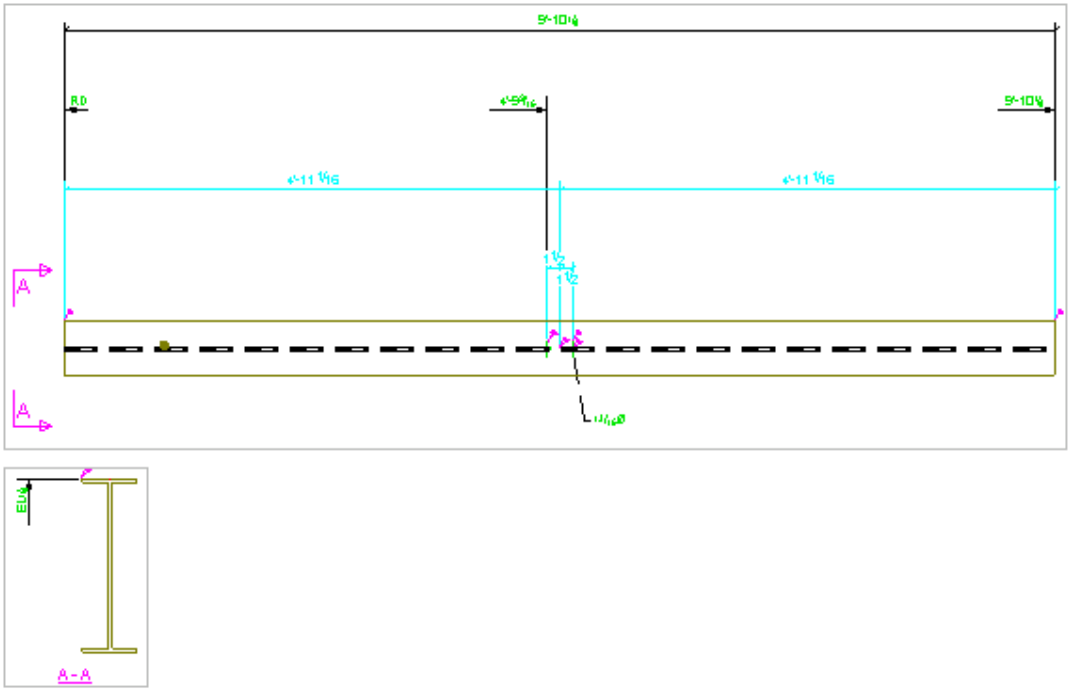
Drawing View

Use to specify the rotation of section views.

`BY_SYMBOL_MAIN_VIEW` (default) uses the orientation of the view that contains the section symbol. This only applies to section views that Tekla Structures creates automatically. Manually-created views have the same rotation as the view they are created from.



`BY_MAIN_VIEW` uses the same orientation as the main view.



This advanced option is model specific and the setting is saved in the options database.

XS_RUN_AT_STARTUP

Category: File Locations

Tekla Structures will automatically launch any executable (`.exe`) files that are located in the folders defined for this advanced option. You can enter several folders separated by a semicolon (;). By default, this advanced option is set as follows:

```
..\<version>\bin\applications\Tekla\ApplicationStartup;..\<version>
\environments\common\extensions\ApplicationStartup\
```

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_RUNPATH

Category

This advanced option must be set in an initialization (.ini) file.

This advanced option is system specific and is read from `teklastructures.ini`. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

This advanced option points to the folder where Tekla Structures searches for models by default. When you open the **New** dialog box, the default model folder is displayed in the **Save in** list.

NOTE This advanced option does not affect the **Open** dialog box.

This advanced option is model specific and the setting is saved in the options database.

Example

```
set XS_RUNPATH=C:\TeklaStructuresModels\
```

XSR_USE_NO_FEET_SEPARATOR

Category

Templates and Symbols

Set this advanced option to `TRUE` to use a space instead of the feet separator in drawing tables and reports, for example, 2 4''1/4. To use the feet separator, use the value `FALSE`. The default value is `FALSE`.

You also need to set `XSR_USE_NO_FEET_SYMBOL`.

This advanced option is model specific and the setting is saved in the options database.

See also

XSR_USE_NO_FEET_SYMBOL (page 395)

XSR_USE_NO_FEET_SYMBOL

Category

Templates and Symbols

Set this advanced option to `TRUE` to omit the feet symbol in drawing tables and reports, for example, 2-4''1/4. If you do not want to omit the feet symbol, use the value `FALSE`. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XSR_USE_NO_INCH_SYMBOL

Category

Templates and Symbols

Set this advanced option to `TRUE` to omit the inch symbol in drawing tables and reports, for example, 2-4 ¼. If you do not want to omit the inch symbol, use the value `FALSE`. The default value is `FALSE`.

You also need to set `XSR_USE_NO_FEET_SYMBOL` and `XSR_USE_NO_FEET_SEPARATOR`.

This advanced option is model specific and the setting is saved in the options database.

See also

XSR_USE_NO_FEET_SEPARATOR (page 395)

XSR_USE_NO_FEET_SYMBOL (page 395)

XSR_USE_ZERO_FEET_VALUE

Category

Templates and Symbols

Set this advanced option to `TRUE` to force Tekla Structures to show zero feet for values less one foot, for example, 0'-6''3/4. If you do not want to show zero feet values, use the value `FALSE`. The default value is `FALSE`.

NOTE Setting this advanced option affects also the following:

- Length values in part and bolt marks
- Length values in templates (template attribute `LENGTH`)

This advanced option is model specific and the setting is saved in the options database.

See also

XSR_BOLT_LENGTH_USE_ONLY_INCHES (page 361)

XSR_USE_ZERO_INCH_FOR_FRACTIONS

Category

Templates and Symbols

Set this advanced option to TRUE to show zero inches for values only containing fractions, for example, 2'-3/4 or 0''1/4. If you do not want to do this, use the value FALSE. The default value is FALSE.

This advanced option is model specific and the setting is saved in the options database.

XSR_USE_ZERO_INCH_VALUE

Category

Templates and Symbols

Set this advanced option to TRUE to show zero inches for values only containing feet and fractions, for example, 2'-0''3/4 or 1/4. If you do not want to do this, use the value FALSE. The default value is FALSE.

This advanced option is model specific and the setting is saved in the options database.

1.18 Advanced options - S

XS_SAVE_WITH_COMMENT

Category

Multi-user

Set this advanced option to TRUE (default) to enable the saving of model revision comments in multi-user models and models shared with Tekla Model Sharing.

XS_SCALE_COPIED_OR_MOVED_OBJECTS_IN_DRAWINGS

Category

Drawing Properties

Use to define the scaling of objects that are copied or moved between drawing views that have different scales.

- To scale the objects according to the view scales, set the advanced option to TRUE.

- To keep the object sizes constant, set the advanced option to `FALSE` (default).

This advanced option is model specific and the setting is saved in the options database.

XS_SCALE_MARKS_TO_FIT_LIMIT

Category

Marking: Parts

Tekla Structures fits part marks near the part they belong to by scaling the text height. Set the minimum scale with this advanced option.

The default value is 1.0. This means that if you do not set the advanced option, Tekla Structures does not scale marks. Tekla Structures scales the text height in steps so that first it tries the scale of 0.9. If the mark does not fit, Tekla Structures scales the mark by 0.8, and so on.

Note that part mark's leader line type has to be either **Try along part** or **Always along part**.

This advanced option is model specific and the setting is saved in the options database.

Example

```
XS_SCALE_MARKS_TO_FIT_LIMIT=0.5
```

XS_SCREW_DIAMOND_WITHOUT_PHI

Category

Marking: Bolts

Set this advanced option to TRUE to prevent Tekla Structures from placing a phi symbol outside the frame of bolt marks (diamond type only). The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_SDNF_CONVERT_PL_PROFILE_TO_PLATE

Category

Export

Set this advanced option to `TRUE` to have SDNF exports convert plate profiles (PL) to contour plates. Works with SDNF versions 2.0 and 3.0. If you do not want to use this setting, Set it to FALSE.The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

XS_SDNF_EXPORT_INCLUDE_GLOBAL_ID

Category

Export

Set this advanced option to `TRUE` to revert to the FrameWorksPlus ID number in SDNF exports. The default value is `FALSE`.

This advanced option only affects version 2.0 SDNF export, but not version 3.0 export.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SDNF_IMPORT_STORE_MEMBER_NUMBER (page 399)

XS_SDNF_IMPORT_MIRROR_SWAP_OFFSETS

Category

Import

Set this advanced option to `TRUE` to swap the end point offsets and start point offsets when an imported part is mirrored already in the SDNF software. If you set this advanced option to `FALSE`, the end point and start point offsets are not swapped. The default value is `TRUE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_SDNF_IMPORT_STORE_MEMBER_NUMBER

Category

Import

Set this advanced option to TRUE to have Tekla Structures store the FrameWorksPlus ID number in SDNF imports. Tekla Structures stores the ID number in the user-defined attribute SDNF_MEMBER_NUMBER. If you do not want to do this, set the option to FALSE. The default value is TRUE.

This advanced option is model specific and the setting is saved in the options database.

For information on exporting the ID number, click the links below.

See also

XS_SDNF_EXPORT_INCLUDE_GLOBAL_ID (page 399)

XS_PML_EXPORT_INCLUDE_GLOBAL_ID (page 346)

XS_SECONDARY_PART_HARDSTAMP

Category

CNC

Set to TRUE to include hard stamps for main parts and any kind of secondary parts in DSTV files. Set to FALSE to create hard stamps only for main parts. The default value is FALSE.

This advanced option is model specific and the setting is saved in the options database.

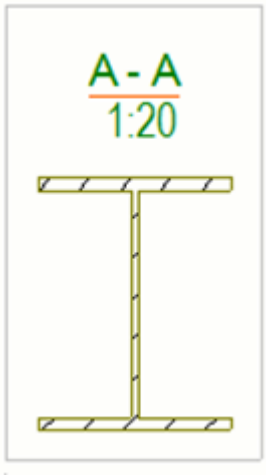
XS_SECTION_LINE_COLOR

Category: Hatching

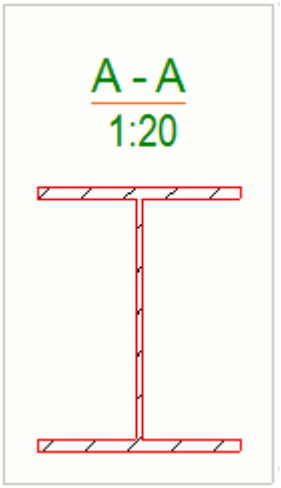
Use to add extra lines in different colors around automatic hatching in section views. Enter the numeric value of the color. See below for the colors and their numeric values.

= 152
= 153
= 160
= 161
= 162
= 163
= 164
= 165
= 154
= 155
= 156
= 157
= 158
= 159
= 130
= 131
= 132
= 133

In the image above, the advanced option has been set to 0:



In the next example, the value 160 is used:



This advanced option is model specific and the setting is saved in the options database.

NOTE To show lines around hatching, and to be able to add extra lines in different colors, you may need to set advanced option `XS_DRAW_ALL_SECTION_EDGES_IN_DRAWINGS` to `TRUE` in the initialization files.

See also

XS_DRAW_ALL_SECTION_EDGES_IN_DRAWINGS (page 179)

XS_SECTION_SYMBOL_LEFT_ARROW_SYMBOL

Category

Drawing Properties

Use to define the custom arrow symbol Tekla Structures uses in section symbols at the left end of sections. To use the custom arrow symbol, select **Custom** from the **Left symbol** list in the **Section symbol properties** dialog box.

By default, Tekla Structures uses the symbol no 1 in the `sections.sym` file (located usually in the folder `\environments\common\symbols\`). If you want to change the symbol, enter the symbol file name first, then the @ sign and then the number of the symbol, for example `sections@1`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SECTION_SYMBOL_RIGHT_ARROW_SYMBOL (page 403)

XS_SECTION_SYMBOL_REFERENCE

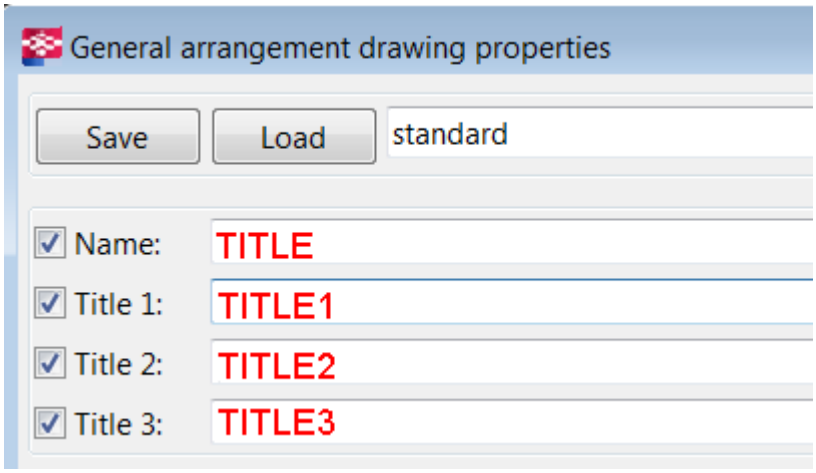
Category

Drawing Properties

Sets the reference text for symbols that show a section in another drawing. The text can include:

- free text
- user-defined attributes
- template attributes

In the **Advanced options** dialog box, use single % characters around the user-defined and template attributes. `%DRAWING_TITLE%` is the default value. `%TITLE%` gives the same result. This advanced option gets the drawing name entered in the drawing properties dialog box. If you enter `TITLE1 - TITLE3`, Tekla Structures gets the drawing title from the drawing properties dialog box. You can also use the format `DR_TITLE1 - DR_TITLE3`.



This advanced option is model specific and the setting is saved in the options database.

XS_SECTION_SYMBOL_RIGHT_ARROW_SYMBOL

Category

Drawing properties

Use to define the custom arrow symbol Tekla Structures uses in section symbols at the right end of sections. To use the custom arrow symbol, select

Custom from the **Right symbol** list in the **Section symbol properties** dialog box

By default Tekla Structures uses the symbol no 0 in the `sections.sym` file (located usually in the folder `\environments\common\symbols\`). Enter the symbol file name first, then the @ sign and then the number of the symbol, for example `sections@0`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SECTION_SYMBOL_LEFT_ARROW_SYMBOL (page 402)

XS_SECTION_VIEW_REFERENCE

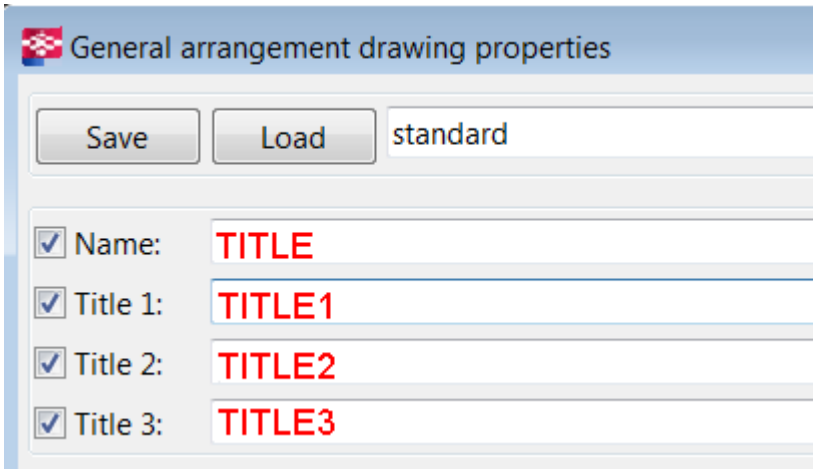
Category

Drawing Properties

Sets the reference text for section view labels. The text can include:

- free text
- user-defined attributes
- template attributes

In the **Advanced options** dialog box, use single % characters around the user-defined and template attributes. `%DRAWING_TITLE%` is the default value. `%TITLE%` gives the same result. This advanced option gets the drawing name entered in the drawing properties dialog box. If you enter `TITLE1 - TITLE3`, Tekla Structures gets the drawing title from the drawing properties dialog box. You can also use the format `DR_TITLE1 - DR_TITLE3`.



This advanced option is model specific and the setting is saved in the options database.

See also

XS_SECTION_SYMBOL_REFERENCE (page 403)

XS_SET_FIXEDMAINVIEW_UDA_TO_AFFECT_NUMBERING

Category: Numbering

To activate the **Show top-in-form face** for non-concrete material, use the advanced option `XS_SET_FIXEDMAINVIEW_UDA_TO_AFFECT_NUMBERING`. The allowed options are `STEEL`, `TIMBER` and `MISC`. You can also combine the options, use a comma (,) as a separator.

This advanced option affects numbering. If parts have different options selected for **Fixed drawing main view**, they get different assembly position numbers.

To show the top-in-form face in drawings and set which view is used in drawings as the main (front) view, go to the user-defined properties of a non-concrete part and select the **Fixed drawing main view** option that you want. The options are **Top**, **Back**, **Bottom**, **Start**, **End** and **Front**.

This advanced option is model specific and the setting is saved in the options database.

XS_SET_HATCH_ORIGIN_INTO_VIEW_ORIGIN

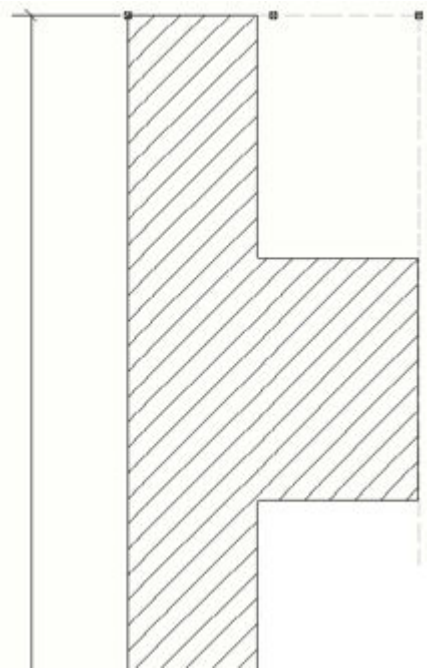
Category: Hatching

Hatches are drawn consistently between parts and set to origin, if you set the advanced option `XS_SET_HATCH_ORIGIN_INTO_VIEW_ORIGIN` to `TRUE` and the scaling and rotation of hatches to zero (0). The default value is `FALSE`.

For automatic hatches, the scale and rotation are set in the `.htc` file, for example, `CONCRETE,0,hardware_LINES,,120`, where the value 0 (or empty) sets auto scale and rotation to false.

For manual hatches, the scale and rotation are set on the **Fill** tab of parts properties or shape properties dialog boxes, where you need to set the **Scale** option to **Custom** and select the **Keep ratio of x and y** check box.

In the example below, the parts that belong to the same cast unit are hatched consistently.



This advanced option is model specific and the setting is saved in the options database.

XS_SET_MAX_POINT_CLOUD_POINT_COUNT

Category: Model View

Use this advanced option to set the default maximum value for the points in a view in point clouds. The default value is 10 000 000 (10 million).

If you discover speed issues, set the count smaller. When the count is smaller, the point cloud appears less dense. If you use a higher value, the dense point cloud may provide more surface kind of appearance due to smaller point distance.

This advanced option is system specific and is read from environment files. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

Restart Tekla Structures after changing the value to activate the new setting.

XS_SHARING_INFO_URL

Category: Multi-user

Use this advanced option to set the Tekla Model Sharing management server address.

This advanced option is system-specific. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

XS_SHARING_JOIN_SHOW_AVAILABLE_UPDATES

NOTE We recommend you to set this advanced option in **File menu** --> **Sharing** --> **Sharing settings**. Select the **Show available updates when joining the model** option.

Use this advanced option to show a list of all the available baselines and updates which to join in Tekla Model Sharing. The list is shown when a user joins a model.

If needed, the advanced option can be set in initialization files. Set `XS_SHARING_JOIN_SHOW_AVAILABLE_UPDATES` to `TRUE` in initialization files to enable the list.

This advanced option is user-specific.

XS_SHARING_READIN_SHOW_AVAILABLE_VERSIONS

NOTE We recommend you to set this advanced option in **File menu** --> **Sharing** --> **Sharing settings**. Select the **Show available updates when reading in the changes** option.

Use this advanced option to show a list of available updates when a user reads in the model changes in Tekla Model Sharing.

If needed, the advanced option can be set in initialization files. Set `XS_SHARING_READIN_SHOW_AVAILABLE_VERSIONS` to `TRUE` in initialization files to enable the list.

This advanced option is user-specific.

XS_SHARING_READIN_SHOW_CHANGEMANAGER

NOTE We recommend you to set this advanced option in **File menu** --> **Sharing** --> **Sharing settings**. Select the **Show changes after read in** option.

Use this advanced option to show a list model changes at the bottom pane after you have read in the model changes in Tekla Model Sharing.

If needed, the advanced option can be set in initialization files. Set `XS_SHARING_READIN_SHOW_CHANGEMANAGER` to `TRUE` in initialization files.

This advanced option is user-specific.

XS_SHARING_READIN_SHOW_CHANGEMANAGER_CONFLICTSONLY

NOTE We recommend you to set this advanced option in **File menu** --> **Sharing** --> **Sharing settings**. Select the **Show changes after read in** and **Only when conflicts exist** options.

Use this advanced option to show a list model changes at the bottom pane after you have read in the model changes in Tekla Model Sharing and when there are conflicts.

If needed, the advanced option can be set in initialization files. Set `XS_SHARING_READIN_SHOW_CHANGEMANAGER` and

XS_SHARING_READIN_SHOW_CHANGEMANAGER_CONFLICTSONLY to TRUE in initialization files.

This advanced option is user-specific.

XS_SHARING_TEMP

Category: Multi-user

Use this advanced option to define the temporary folder for Tekla Model Sharing packet management. The default is the Windows temporary folder.

Note that the folder path can be maximum 70 characters long.

This advanced option is user specific and the setting is saved in options.bin under the user folder, for example, in C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings. Restart Tekla Structures to activate the new value.

XS_SHOP_BOLT_MARK_STRING_FOR_SIZE

Category

Marking: Bolts

Use to define the contents of the **Size** element in bolt marks (workshop). The default value is %BOLT_NUMBER%*D%HOLE.DIAMETER% - M%DIAMETER%x%LENGTH%.

This advanced option is only used when there is a bolt and the hole is a normal one.

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches %%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X

- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)
- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD
- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK
- BOLT_SHORT_NAME
- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SHOP_BOLT_MARK_STRING_FOR_SIZE_IN_GA

Category

Marking: Bolts

Use to define the contents of the size element in bolt marks for workshop bolts in general arrangement drawings.

This advanced option is only used when there is a bolt and the hole is a normal one.

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)

- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD
- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK
- BOLT_SHORT_NAME
- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SHOP_HOLE_MARK_STRING_FOR_SIZE_IN_GA (page 412)

XS_SHOP_LONGHOLE_MARK_STRING_FOR_SIZE_IN_GA (page 414)

XS_SHOP_HOLE_MARK_STRING_FOR_SIZE

Category

Marking: Bolts

Use to define the contents of the **Size** element in hole marks (workshop). For example, to show bolt number and hole diameter, enter `%BOLT_NUMBER%*D %HOLE.DIAMETER%`.

This advanced option is only used when there is a hole, no bolt (and the hole is a normal one).

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)

- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD
- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK
- BOLT_SHORT_NAME
- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SHOP_HOLE_MARK_STRING_FOR_SIZE_IN_GA

Category

Marking: Bolts

Use to define the contents of the size element in hole marks for workshop bolts in general arrangement drawings.

This advanced option is only used when there is a hole, no bolt (and the hole is a normal one).

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)
- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD

- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK
- BOLT_SHORT_NAME
- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SHOP_BOLT_MARK_STRING_FOR_SIZE_IN_GA (page 410)

XS_SHOP_LONGHOLE_MARK_STRING_FOR_SIZE_IN_GA (page 414)

XS_SHOP_LONGHOLE_MARK_STRING_FOR_SIZE

Category

Marking: Bolts

Use to define the contents of the **Size** element in slotted hole marks (workshop). The default value is `%BOLT_NUMBER%*D%HOLE.DIAMETER% (%HOLE.DIAMETER+LONG_HOLE_X%x%HOLE.DIAMETER+LONG_HOLE_Y%) - M %DIAMETER%x%LENGTH%`.

This advanced option is only used when there is a longhole.

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)
- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD

- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK
- BOLT_SHORT_NAME
- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SHOP_LONGHOLE_MARK_STRING_FOR_SIZE_IN_GA

Category

Marking: Bolts

Use to define the contents of the size element in longhole marks for workshop bolts in general arrangement drawings.

This advanced option is only used when there is a longhole.

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)
- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD
- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK

- BOLT_SHORT_NAME
- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SHORTENING_SYMBOL_COLOR

Category

Drawing Properties

Use this advanced option to set the color of the view shortening symbol. Enter an integer value. Default is the same as the part color. See below for integers for different colors.

This advanced option is model specific and the setting is saved in the options database.

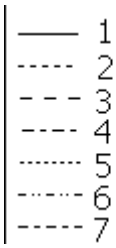
See also

XS_SHORTENING_SYMBOL_LINE_TYPE

Category

Drawing Properties

Use this advanced option to set the line type of the view shortening symbol. Enter an integer value. Default is a solid line. See below for integer values for different line types.



This advanced option is model specific and the setting is saved in the options database.

See also

XS_SHORTENING_SYMBOL_WITH_ZIGZAG

Category

Drawing Properties

Use this advanced option to set the shortening symbol line shape. If you do not want to use zigzag, set this advanced option to `FALSE`. `TRUE` is the default.

This advanced option is model specific and the setting is saved in the options database.

TIP You can control the appearance of the view shortening symbol with the advanced options XS_SHORTENING_SYMBOL_COLOR (page 415) and XS_SHORTENING_SYMBOL_LINE_TYPE (page 415).

To use a view shortening symbol instead of the empty area, set the advanced options XS_DRAW_VERTICAL_VIEW_SHORTENING_SYMBOLS_TO_PARTS (page 199) and XS_DRAW_HORIZONTAL_VIEW_SHORTENING_SYMBOLS_TO_PARTS (page 193) to `TRUE`.

See also

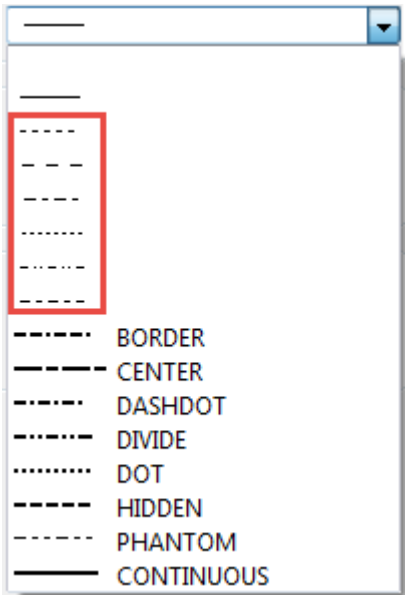
XS_SHOW_HARDWARE_DASHED_LINE_IN_PIXEL_SCALE

Category: **Drawing Properties**

Set `XS_SHOW_HARDWARE_DASHED_LINE_IN_PIXEL_SCALE` to `FALSE` to activate the functionality, where dashed hardware line scale is the same as in printed and exported drawings, and the dashed hardware lines work in the same way as the custom lines.

Dashed hardware line types are scaled by pixels by default (`TRUE`), which means that they look the same in all zoom levels.

Dashed hardware lines are the first couple of lines in the line list:



This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XSR_SHOW_INCH_MARK_IN_PROFILE_NAMES

Category

Templates and Symbols

Use to show or hide the inch symbol in the profile name in reports and templates. When you set the advanced option to `TRUE`, the profile appears like this: PL2 1/2"X20". When you set the advanced option to `FALSE`, the profile appears like this: PL2 1/2X20. The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_PROFILE_DISPLAY_INCH_MARK_AFTER_FRACTIONS_IN_REPORTS (page 359)

XS_SHOW_NOTIFICATION_REPORT

Category: Modeling Properties

Set to `TRUE` to run and display a notification report from the whole model when you open a model.

You can use the notification report to, for example:

- List assignments on all objects (drawings, parts and assemblies).
- Report how many assemblies there are in the model, how many assembly drawings have been created, and how many of them have been approved and how many have been issued for production.

The report template used is named `notification_report`, and you can edit it in Template Editor. Notification report reports anything you want, not just assignments. You need to edit the report template to report the things you need. For example, some environments do not have all attributes available, like `ASSIGNED_TO` or `ASSIGNED_BY`, for parts and assemblies.

If you do not want to display the notification report upon opening a model, set the advanced option to `FALSE`, which is the default value.

This advanced option is model specific and the setting is saved in the options database.

Example of a list of drawing assignments:

```
 - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
PROJECT NEWS!!!!

This is an automatic newsflash for project participants

This message is shown when anyone opens the model or runs "notification_report" report template
 - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
-------------------------------------------------------------------------
NOTIFICATION OF ASSIGNMENTS CONTRACT NO:   1            Page: 1
Objects assigned to user1                              Date: 07.11.2016
-------------------------------------------------------------------------
Object         Name        Assigned to    Assigned by       Comment
-------------------------------------------------------------------------
A Drawing     [A.1]  PLATE        user1          user2       OK
A Drawing     [C.2]  COLUMN       user1          user2       OK
-------------------------------------------------------------------------
0 assemblies, 0 parts and 2 drawings have been assigned to
-------------------------------------------------------------------------
```

XS_SHOW_PERFORM_NUMBERING_MESSAGE

Category

Numbering

Use to show or hide the **Perform numbering** button in the warning message Tekla Structures displays if you try to create a drawing without numbering or when numbering is not up to date.

- By default, this advanced option is set to `TRUE` and the numbering button is shown.
- To hide the button, set this advanced option to `FALSE`.

Consider hiding the numbering button in the multi-user mode, because unnecessary or unplanned numbering may take a while in large models or break project numbering. This can occur if **Synchronize with master model (save-numbering-save)** check box is not selected in the **Numbering setup** dialog box.

This advanced option is model specific and the setting is saved in the options database.

XS_SHOW_PROGRESS_BAR_FOR_PROJECT_STATUS_VISUALIZATION

Category

Model View

Use for defining whether the progress bar for project status visualization is displayed or not.

To display the progress bar, set this advanced option to `TRUE` (default). If you do not want to display the progress bar, set this advanced option to `FALSE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_SHOW_REVISION_MARK_ON_DRAWING_LIST

Category: Drawing Properties

Set this advanced option to `TRUE` to have Tekla Structures include the **Revision mark** in **Document manager** instead of the **Revision number**. The default value is `FALSE`. This means that the revision number is shown.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_SHOW_SHADOW_FOR_ORTHO_IN_DX

Category: **Model View**

Use this advanced option to control whether shadows are shown in the DirectX rendered model views when the orthogonal projection is on. Shadows are more noticeable in the perspective projection than in the orthogonal projection.

By default, this advanced option is set to `FALSE`.

If you change the value, you need to reopen the view to activate the new value.

XS_SHOW_SHADOW_FOR_PERSPECTIVE_IN_DX

Category: **Model View**

Use this advanced option to control whether shadows are shown in the DirectX rendered model views when the perspective projection is on. Shadows are more noticeable in the perspective projection than in the orthogonal projection.

By default, this advanced option is set to `TRUE`.

If you change the value, you need to reopen the view to activate the new value.

XS_SHOW_SITE_STUDS_IN_ASSEMBLY_DRAWINGS

Category

Drawing Properties

Set this advanced option to `TRUE` (default) to show site studs in assembly drawings. If you set it to `FALSE`, the site studs are not shown.

This advanced option is model specific and the setting is saved in the options database.

XS_SHOW_STATISTICS_IN_DX

Category: Model View

Use this advanced option to show statistics and the used rendering device in the DirectX rendered model views. The statistics are shown in the lower left corner of the model view, as frames per second measurement.

With this advanced option you can easily verify that you are using a proper graphics hardware accelerator for the Tekla Structures model views, especially in the case of multiple GPUs such as laptops, which often have both CPU built accelerator and an external, much more powerful graphics accelerator

By default, the advanced option is set to `FALSE`.

If you change the value, you need to reopen the view to activate the new value.

Note that this advanced option has no effect on the legacy OpenGL rendered model views.

XS_SHOW_STUDS_IN_WORKSHOP_DRAWINGS

Category

Drawing Properties

Set this advanced option to `TRUE` to show workshop studs in single-part drawings. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_SHOW_TEMPLATE_LOG_MESSAGES

Category

Templates and Symbols

Set this advanced option to `TRUE` to show error messages related to template attributes in the log file. Set it to `FALSE` to hide these messages. By default this advanced option is set to `FALSE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_SINGLE_CENTERED_SCREW

Category

Single Part View in Assembly Drawing

Use to control the dimensioning of centrally-located bolts in single-part views included in assembly drawings. Possible values are:

- 0 = Dimensions the centered bolts spread.
- 5 = Dimensions the bolts to the main part center-lines.
- 6 = Overrides bolt overrides the **Secondary part bolt internal dimensions** setting for centered bolts. This only applies to bolts located centrally on the part.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SINGLE_CLOSE_DIMENSIONS

Category

Single part view in assembly drawing

Use to close dimensions in single-part views included in an assembly drawing.

- 0 = Does not close dimensions
- 1 = Closes dimensions in the x direction and leaves others open. This is the default value.
- 2 = Closes all dimensions

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SINGLE_CLOSE_SHORT_DIMENSIONS

Category

Single Part View in Assembly Drawing

Use to close short dimensions in single-part views included in assembly drawings. The default value is `1`. If you do not want to close short dimensions, enter `0`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SINGLE_COMBINE_DISTANCE

Category

Single Part View in Assembly Drawing

Use to set a distance for combining dimensions in single-part views included in assembly drawings. Enter a decimal value, for example `400.0`.

By default, this advanced option is not set to any value.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SINGLE_COMBINE_MIN_DISTANCE

Category

Single Part View in Assembly Drawing

Use to set a minimum distance for combining dimensions in single-part views included in assembly drawings. Enter a decimal value, for example `200.0`.

By default, this advanced option is not set to any value.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SINGLE_COMBINE_WAY

Category

Single Part view in assembly drawing

Use to combine dimensions in single-part views included in assembly drawings. The options correspond to the combining order on the General tab of the **Dimensioning Properties** dialog box. By default, this advanced option is not set to any value.

- Option `0` prevents dimensions from being combined.
- Option `1` combines part position dimensions with part internal dimensions, and bolt group internal dimensions with bolt edge distances. Bolt position dimensions are not combined with bolt internal dimensions.
- Option `2` combines the part position dimension with part internal dimensions and bolt group internal dimensions. Bolt internal dimensions are combined with bolt position dimensions. Edge distances are shown separately.
- Option `3` combines bolt internal dimensions and position dimensions in the same dimension line.
- Option `4` combines bolt group position dimensions with part position dimensions. Part and bolt internal dimensions are not combined with this option, but bolt internal dimensions are combined with bolt edge distances.
- Option `5` combines internal dimensions and the position dimension of bolt groups where there are several bolt groups.
- Option `4.5` uses a combination of option `5` for the main part and a combination of option `4` for the secondary parts.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SINGLE_DIMENSION_TYPE

Category

Single Part View in Assembly Drawing

Use to set a dimension type for single-part views included in assembly drawings. The options are:

- 1 = Relative, point to point dimensions. This is the default value.
- 2 = Absolute, dimensions from a common start point.
- 3 = Relative and absolute, a combination of point to point and common start point.
- 4 = US absolute, dimensions from a common start point, which include a running dimension mark (RD).

- 16 = US absolute 2, similar to US absolute, but it changes short dimensions to relative.
- 35 = Absolute plus short relatives, which is similar to Absolute, but it changes short dimensions to relative. Also called internal absolute. This option may show both dimensions, but it does not show relative dimensions when dimensions are long. This option shows the absolute dimensions inside the dimension lines.
- 99 = Absolute plus all relatives above the absolutes, which is similar to Relative and absolute, but it places the relative dimensions above the absolute.

This advanced option is model specific and the setting is saved in the options database.

XS_SINGLE_DRAW_PART_AS

Category

Single Part View in Assembly Drawing

Use to define how parts appear (their representation type) in single-part views included in assembly drawings. The options are:

- `1` (default) solid
- `4` workshop solid (round tubes open)
- `2` symbol form

This advanced option is model specific and the setting is saved in the options database.

XS_SINGLE_EXCLUDE

Category

Single Part View in Assembly Drawing

Use to exclude single-part views from assembly drawings. The options are:

- `NONE` includes all single part views.
- `MAIN_SHAFT` includes single part views of all parts, except assembly main parts.
- `ALL_SHAFTS` includes single part views of all parts, except those with another object welded to them (i.e. main parts).
- `AUTO` (default) includes single part views of all parts, except the longest main part in the assembly.

- `ALL_BUT_MAIN_PART` includes single part views of assembly main parts only.
- `STANDARD` includes single-part views of all parts, except standard parts. You can add `STANDARD` after any of the existing values, for example `MAIN_SHAFT_AND_STANDARD` creates single-part drawing from all but main part or standard parts.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SINGLE_FORWARD_OFFSET

Category

Single Part View in Assembly Drawing

Use to set the distance Tekla Structures uses to search for the base point of a dimension in single-part views included in an assembly drawing. If Tekla Structures does not find a base point (corner) within the defined forward offset search distance, it uses an edge point. Enter the value as a decimal, for example `250.0`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SINGLE_NO_RELATIVE_SHAPE_DIMENSIONS

Category

Dimensioning: Parts

Set this advanced option to `TRUE` (default) to force shape dimensions to be the same as the dimension type you select. If you do not want to do this, set it to `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_NO_RELATIVE_SHAPE_DIMENSIONS (page 327)

XS_SINGLE_NO_SHORTEN

Category

Single part view in assembly drawing

Set this advanced option to `TRUE` to display single-part views in drawings without shortening parts. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_SINGLE_ORIENTATION_MARK

Category: Single Part View in Assembly Drawing

You can display orientation symbols in single-part views included in assembly drawings. To show the orientation symbols in single-part views:

1. Click **File menu** --> **Settings** --> **Advanced options** and go to the **Single Part View in Assembly Drawing** category.
2. Enter `TRUE` as the value.
3. In assembly drawing properties, click **Layout** and go to the **Other** tab.
4. Set the option **Single-part attributes** to **Current attributes**. Tekla Structures takes the orientation mark settings from the current single-part drawing properties. If you select other attributes, the visibility of the orientation symbols is set according to the selected attribute file.

The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_SINGLE_PART_DRAWING_VIEW_TITLE

Category: **Drawing Properties**

Use to define a title for single-part drawing views in multidrawings. You can use any of the following to define the title:

Enter any combination of text and options, and enclose each option in percent symbols (%).

Example: `No. %MODEL_NUMBER%, Marked %PART_POS%`

The options are:

- `PART_NAME`
- `PART_MATERIAL`

- PART_POS
- ASSEMBLY_POS
- MODEL_NUMBER
- LENGTH
- PROFILE
- MAIN_PART_LENGTH
- DRAWING_BASE_NAME
- DRAWING_NAME

This advanced option is model specific and the setting is saved in the options database.

XS_SINGLE_PART_EXTREMA

Category

Single Part View in Assembly Drawing

Use to display overall dimensions in single-part views included in assembly drawings. Enter one of the following values:

- 0 = None
- 2 = Once
- 3 = All

This advanced option is model specific and the setting is saved in the options database.

XS_SINGLE_PART_SHAPE

Category

Single Part View in Assembly Drawing

Use to show or hide shape dimensions in single part views.

Set to `1` (default) to show shape dimensions in single-part views.

Set to `0` to hide shape dimensions in single-part views.

Tekla Structures automatically creates radius dimensions for curved chamfers in single-part drawings when you use shape dimensions.

This advanced option is model specific and the setting is saved in the options database.

XS_SINGLE_SCALE

Category

Single Part view in Assembly Drawing

Use to set the scale of single-part views included in assembly drawings. Enter a decimal value. By default, this advanced option is not set to any value.

This advanced option is model specific and the setting is saved in the options database.

Example

To have Tekla Structures use the scale 1/10, enter `10.0`.

This advanced option is related to the advanced option XS_USE_EXISTING_SINGLE_PART_DRAWINGS_SCALE (page 468).

XS_SINGLE_SCREW_INTERNAL

Category

Single Part View in Assembly Drawing

Use to show or hide internal bolt dimensions in single part views included in assembly drawings.

Set to `1` to show internal bolt dimensions.

Set to `0` to hide internal bolt dimensions.

By default, this advanced option is not set to any value.

This advanced option is model specific and the setting is saved in the options database.

See also

Dimensioning properties - Bolt dimensions tab (Integrated dimensioning) (page 725)

XS_SINGLE_SCREW_POSITIONS

Category

Single Part View in Assembly Drawing

Use to include position dimensions for bolts in single-part views included in assembly drawings. Enter one of the following values:

- 0 = Off

- 1 = On

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SINGLE_USE_WORKING_POINTS

Category

Single Part View in Assembly Drawing

Use to include dimensions from work points in single-part views included in assembly drawings. Enter one of the following values:

- 0 = None (default)
- 1 = Main part
- 2 = Working points
- 3 = Both

This advanced option is model specific and the setting is saved in the options database.

XS_SINGLE_X_DIMENSION_TYPE

Category

Single Part View in Assembly Drawing

Use to set the dimension type for single-part views included in assembly drawings. These are otherwise like straight dimensions set with the advanced option `XS_SINGLE_X_DIMENSION_TYPE` but they override the straight setting for horizontal dimensions.

- 0 = Tekla Structures uses straight dimension settings
- 1 = Relative, point to point dimensions. This is the default value.
- 2 = Absolute, dimensions from a common start point.
- 3 = Relative and absolute, a combination of point to point and common start point.
- 4 = US absolute, dimensions from a common start point, which include a running dimension mark (RD).
- 16 = US absolute 2, similar to US absolute, but it changes short dimensions to relative.

- 35 = Absolute plus short relatives, which is similar to Absolute, but it changes short dimensions to relative. Also called internal absolute. This option may show both dimensions, but it does not show relative dimensions when dimensions are long. This option shows the absolute dimensions inside the dimension lines.
- 99 = Absolute plus all relatives above the absolutes, which is similar to Relative and absolute, but it places the relative dimensions above the absolute.

This advanced option is model specific and the setting is saved in the options database.

XS_SITE_BOLT_MARK_STRING_FOR_SIZE

Category

Marking: Bolts

Use to define the contents of the **Size** element in bolt marks (site). The default value is `%BOLT_NUMBER%*D%HOLE.DIAMETER%`.

This advanced option is only used when there is a bolt and the hole is a normal one.

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)
- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD
- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK

- BOLT_SHORT_NAME
- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SITE_BOLT_MARK_STRING_FOR_SIZE_IN_GA

Category

Marking: Bolts

Use to define the contents of the size element in bolt marks for site bolts in general arrangement drawings.

This advanced option is only used when there is a bolt and the hole is a normal one.

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)
- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD
- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK
- BOLT_SHORT_NAME

- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SITE_HOLE_MARK_STRING_FOR_SIZE_IN_GA (page 434)

XS_SITE_LONGHOLE_MARK_STRING_FOR_SIZE_IN_GA (page 436)

XS_SITE_HOLE_MARK_STRING_FOR_SIZE

Category

Marking: Bolts

Use to define the contents of the **Size** element in hole marks (site). For example, to show bolt number and hole diameter in the mark, enter `%BOLT_NUMBER%*D%HOLE.DIAMETER%`.

This advanced option is only used when there is a hole, no bolt (and the hole is a normal one).

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)
- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD
- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK
- BOLT_SHORT_NAME

- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SITE_HOLE_MARK_STRING_FOR_SIZE_IN_GA

Category

Marking: Bolts

Use to define the contents of the size element in hole marks for site bolts in general arrangement drawings.

This advanced option is only used when there is a hole, no bolt (and the hole is a normal one).

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)
- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD
- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK
- BOLT_SHORT_NAME

- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SITE_BOLT_MARK_STRING_FOR_SIZE_IN_GA (page 432)

XS_SITE_LONGHOLE_MARK_STRING_FOR_SIZE_IN_GA (page 436)

XS_SITE_LONGHOLE_MARK_STRING_FOR_SIZE

Category

Marking: Bolts

Use to define the contents of the **Size** element in slotted hole marks (site). The default value is `%BOLT_NUMBER%*D%HOLE.DIAMETER%(%HOLE.DIAMETER +LONG_HOLE_X%x%HOLE.DIAMETER+LONG_HOLE_Y%)`.

This advanced option is only used when there is a longhole.

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)
- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD
- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK
- BOLT_SHORT_NAME

- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SITE_LONGHOLE_MARK_STRING_FOR_SIZE_IN_GA

Category

Marking: Bolts

Use to define the contents of the size element in longhole marks for site bolts in general arrangement drawings.

This advanced option is only used when there is a longhole.

You can use any combination of text and the following options as the value for this advanced option. Enclose each option in % characters.

Note that if you are defining this advanced in an .ini file, use double percent signs around the switches `%%BOLT_NUMBER%%*D%%HOLE.DIAMETER%%`.

To use special characters enter a backslash (\) followed by an ASCII number. You can use the options in any order, and make calculations.

- BOLT_NUMBER
- DIAMETER
- LENGTH
- HOLE.DIAMETER
- LONG_HOLE_X
- LONG_HOLE_Y
- LONGHOLE_MIN (the shorter of the slotted hole dimensions)
- LONGHOLE_MAX (the longer of the slotted hole dimensions)
- BOLT_STANDARD
- BOLT_MATERIAL
- BOLT_ASSEMBLY_TYPE
- BOLT_COUNTERSUNK
- BOLT_SHORT_NAME

- BOLT_FULL_NAME

This advanced option is model specific and the setting is saved in the options database.

See also

XS_SITE_HOLE_MARK_STRING_FOR_SIZE_IN_GA (page 434)

XS_SITE_BOLT_MARK_STRING_FOR_SIZE_IN_GA (page 432)

XS_SKIP_START_UP_SIGNIN_ON_PREMISE_LICENSING

This advanced option `XS_SKIP_START_UP_SIGNIN_ON_PREMISE_LICENSING` skips the sign in dialog box during the Tekla Structures start-up when you set it to `TRUE`. It is especially useful if you use on-premises licenses and do not want to or cannot sign in.

This advanced option must be read by Tekla Structures in the early stages of starting up. You can set it as a Windows environment variable, in a batch file, or in bypass.ini.

XS_SNAPSHOT_DIRECTORY

Category: File Locations

Use this advanced option to define the folder where Tekla Structures stores screenshots when you go to the **View** tab (**Views** in drawings) and click **Screenshot** --> **Screenshot** . If you do not define a path, Tekla Structures stores screenshots in the current model folder. The default value is `.\screenshots\`.

If the defined folder does not exist, Tekla Structures automatically creates it when you take a screenshot.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

Example

```
c:\temp\
```

XS_SOLID_BUFFER_SIZE

Category: Speed and Accuracy

Use this advanced option to define the size of the solid object buffer. The value is the number of simple average solids. The buffer size setting depends on your environment.

Tekla Structures creates a buffer in system RAM to store the solid representation of parts it creates during certain processes. For example, Tekla Structures creates solid objects when numbering a model. When this buffer fills up, Tekla Structures erases the contents in order to continue using the buffer.

When you increase the buffer size, Tekla Structures keeps more solid objects in the memory, and does not have to recreate them so often. Increasing this value also increases the memory used by the processes. Decreasing this value decreases the memory requirements, but also the performance as Tekla Structures has to recreate solid objects more often.

Testing various settings is the best way to optimize the solid object buffer size. Remember that opening Tekla Structures windows uses some RAM, and that other programs also use RAM for various processes.

For optimum performance for large models under restricted memory conditions, consider having the solid cache size in between 0.2 - 5 * the number of parts in your model. To find out the number of parts in the model, go to the **Edit** tab and click **Inquire** --> **Model size** .

In 64-bit machines you do not usually have to change the default value. If the amount of memory is large, keep the buffer size at least as large as the number of parts in the model.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_SOLID_USE_HIGHER_ACCURACY

This advanced option must be set in an initialization (.ini) file.

NOTE This advanced option increases the number of faces in solid objects, which slows down Tekla Structures. We recommend using this advanced option only when needed.

When you set this advanced option to `TRUE` in the `options.ini` file in the model folder:

- When you have set part representation to **Exact** in the **Display** dialog box, parts are shown with roundings. Also weld polygons include profile roundings.
- When the advanced option XS_DISPLAY_FILLET_EDGES (page 170) is set to TRUE, lines are shown between the roundings and straight segments of parts.
- Part volume, area and net weight is and are closer to real manufactured values.

Set this advanced option to `TRUE` in the `options.ini` file in model folder. The default value is `FALSE`.

Limitations

- Do not activate this advanced option when you create NC/DSTV files, because it can lead to inaccuracies in the exported data.
- Do not activate this advanced option when you create single part or assembly drawings, because it can lead to unnecessary dimension line creation.
- In components, some stiffeners might bite into flanges of columns or beams.
- Some bolt edge distances are not correctly calculated.

See also

XS_CS_CHAMFER_DIVIDE_ANGLE (page 139)

XS_STACKED_FRACTION_TYPE

Category

Imperial units

Use to define the appearance of fractions. You can use stacked fractions in drawings, text, marks etc., but not in templates. The options are (from left to right in the illustration):

- `NOT_STACKED`
- `DASH`
- `SLASH`
- `WITHOUT_SLASH`

If you do not want to use stacked fractions, set this advanced option to `NOT_STACKED`, (default) or use a backslash character (\ ) before the slash character (/) in the text (e.g. 1\ /16).

This advanced option is model specific and the setting is saved in the options database.

XS_STANDARD_GUSSET_WIDTH_TOLERANCE

Category

Components

Use to define the tolerance between the actual and standard widths of gusset plates created using modeling tools or connections. Tekla Structures uses a standard plate file to define the default plate width including the tolerance value. Enter the value in millimeters, for example 1.0.

Components that use this advanced option are:

- **Welded gusset (10)**
- **Bolted gusset (11)**
- **Tube gusset (20)**
- **Corner tube gusset (56)**
- **Corner bolted gusset (57)**
- **Wraparound gusset (58)**
- **Hollow brace wraparound gusset (59)**
- **Wraparound gusset cross (60)**
- **Gusseted cross (62)**
- **Corner wrapped gusset (63)**

This advanced option is model specific and the setting is saved in the options database.

XS_STANDARD_STIFFENER_WIDTH_TOLERANCE

Category: Components

Use to define the tolerance between the actual and standard widths of stiffener plates created by **Stiffened shear plate (17)**, **Haunch (40)**, **Beam with stiffeners (129)**, **Column with shear plate (131)**, **Bolted moment connection (134)**, **Beam to beam stub (135)**, **Clip angle (141)**, **Bent plate (151)**, **Moment connection (181)**, **Column with stiffeners W (182)**, **Column**

with stiffeners (186), **Column with stiffeners S (187)**, **Stiffeners (1003)**, **Stub (28)**, **Web stiffened base plate (1016)**, and **Multiple stiffeners (1064)**.

Enter the value in millimeters. Do not use the value `0`. The advanced option is set to `10` by default.

This advanced option is model specific and the setting is saved in the options database.

XS_STD_LOCALE

Category

This advanced option is system specific and should be set in the `lang_<CurrentLanguage>.ini` files.

Use this advanced option to be able to open drawings in a situation where you have English Tekla Structures and multi-byte locale Windows operating system. Set it to one of the following values in the `teklastructures.ini` file, depending on the locale of your operating system:

- `set XS_STD_LOCALE=japanese`
- `set XS_STD_LOCALE=chinese-traditional`
- `set XS_STD_LOCALE=chinese-simplified`
- `set XS_STD_LOCALE=russian_us.1251`
- `set XS_STD_LOCALE=korean_korea.949`

If you set `XS_STD_LOCALE` to any other value, or leave the value out, English locale is used by default.

NOTE For more information on locales, see https://msdn.microsoft.com/en-us/library/ms903928.aspx.

XS_STD_PART_MODEL

Category

Numbering

To use standard parts in numbering, enter the standard part model folder path. A standard-part model contains only standard parts with specific part prefixes. As Tekla Structures carries out the numbering, it compares all of the parts in the current model to the standard-part model. The numbering applies any part position numbers (only the part prefix) found in the standard-part

model to all identical parts found in the current model.To not use standard parts, leave the value out.

This advanced option is model specific and the setting is saved in the options database.

Example

```
XS_STD_PART_MODEL=C:\TeklaStructuresModels\StandardParts\
```

See also

General numbering settings (page 629)

XS_STEEL1_TS_PAGE_9_EXTENSION

Category

This advanced option must be set in an initialization (.ini) file.

Used for localizing the contents of Component page 9. By default standard component settings are used. For example, in the USA environment, you can use the value `_usimp`.

XS_STEEL1_TS_PAGE_10_EXTENSION

Category

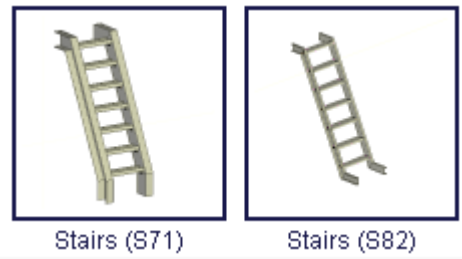
This advanced option must be set in an initialization (.ini) file.

Used for localizing the contents of Component page 10 (catalog steps in stair components). By default standard component settings are used.

Step profiles are coming from `steps.dat` file, which is environment specific. To get a profile visible in the catalogue step/step profile list in the **Stairs** component dialog box, the names of the profiles have to be listed also in the environment-specific `.inp` file in the `\TeklaStructures\<version>\applications\stee11` folder. There are different files for each environment: `ts_page_10_australasia.inp`, `ts_page_10_austria.inp`, `ts_page_10_china.inp`, and so on. For example, in the USA environment, you can use the value `_usimp` for this advanced option.

Affected stair components and options:

- **Stairs (S71)**: Stair setup tab: Catalogue step
- **Stairs (S82)**: Parameters tab: Step profile



For Stairs S71, the **Step type** must be set to **Catalogue step**, to be able to select the step profile from the **Catalogue step** list.

XS_STORE_MULTIPLE_BAK_FILES

Category

File Locations

Set this advanced option to `TRUE` to store multiple versions of the backup copy of the model database. The default value is `FALSE`.

A new copy of the `.bak` backup file is created each time the model is saved. The name of each backup file includes the date and time the file was created. Old or unnecessary files need to be deleted manually.

XS_SUPERSCRIPT_HEIGHT_FACTOR

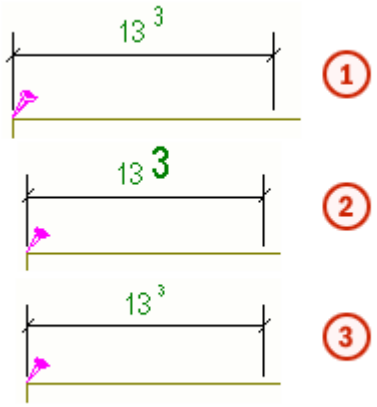
Category

Dimensioning: General

Use this advanced option to set the scale factor for the text height in superscripts used in dimensions. The default is `0.7`.

This advanced option is model specific and the setting is saved in the options database.

Examples



Example number	The advanced option is set to
1	0.7
2	1.5
3	0.5

XS_SUPERSCRIPT_USED_IN_DRAWING_TEXTS

Category

Dimensioning: General

Set to TRUE to enable the showing of superscript in texts in drawings, and to FALSE to disable it. The default is TRUE.

This advanced option is model specific and the setting is saved in the options database.

XS_SWITCH_MULTI_NUMBERS_FOR

Category: Numbering

Use to define the order in which multidrawing numbers, and part or assembly numbers appear in multi-numbers. The options are: NONE, ASSEMBLIES,

PARTS and ASSEMBLIES_AND_PARTS. The default value is ASSEMBLIES_AND_PARTS.

This advanced option is model specific and the setting is saved in the options database.

Example

If this advanced option is set to PARTS, the presentation of multinumbers for parts is 101a, not a101.

See also

XS_USE_MULTI_NUMBERING_FOR (page 472)

XS_SWITCH_POS_NUMBERS_FOR

Category

Numbering

Changes the prefix you define for Tekla Structures assembly and/or part marks to a suffix (for example, A1 becomes 1A). The options are NONE, PARTS, ASSEMBLIES, and ASSEMBLIES_AND_PARTS. The default value is ASSEMBLIES_AND_PARTS.

NOTE The advanced options XS_ASSEMBLY_POSITION_NUMBER_FORMAT_STRING (page 87) and XS_PART_POSITION_NUMBER_FORMAT_STRING (page 340) override the advanced option XS_SWITCH_POS_NUMBERS_FOR. XS_SWITCH_POS_NUMBERS_FOR has no impact on assembly and/or part position numbers if you use XS_ASSEMBLY_POSITION_NUMBER_FORMAT_STRING and/or XS_PART_POSITION_NUMBER_FORMAT_STRING.

This advanced option is model specific and the setting is saved in the options database.

SYMEDHOME

Category: Templates and Symbols

Points to the location of the Symbol Editor program file symed.exe. The default value is %XSBIN%.

This advanced option is system specific and is read from environment files. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

XS_SYSTEM

Category: File locations

NOTE This advanced option is only meant for administrators.

It points to the location of the Tekla Structures system folder. The system folder is used for storing files that define default settings. These include the standard, data (`.dat`), property files, drawing files, template files, and report files, for example.

Specifying more than one system folder

You can specify more than one system folder, and this way define specific settings for each role. Use the role options defined in the `env_<environment>.ini` file to point to the roles when specifying the system folders in `XS_SYSTEM`. For example, `XS_STEEL` (`\Steel`), `XS_CONCRETE` (`\Concrete`), `XS_ENGINEERING` (`\Engineering`) and `XS_PRECAST` (`\Precast`) each point to the folders that contain the settings specific to that role. An example for steel role option in the `env_<environment>.ini` file could be as follows:

```
set XS_STEEL=%XSDATADIR%\environments\Steel\master_drawings\;%XSDATADIR%
\environments\Steel\model_filters\;%XSDATADIR%\environments\Steel
\model_settings\
```

To specify several system folders in `XS_SYSTEM`, enter the options pointing to the role options and separate them by semicolons.

Example

```
set XS_SYSTEM=%XS_STEEL%;%XS_ENGINEERING%;%XS_CONTRACTOR%;%XS_GENERAL
%;%XSDATADIR%\environments\common\system\
```

Tekla Structures searches the folders from right to left. If files with identical names exist in several folders, the one that is read last will be used. In the example above, the files found from `XS_STEEL` will be used instead of the files with identical names in `common\system\` or in any other folder mentioned before the last folder.

This is a system-specific advanced option and cannot be changed.

1.19 Advanced options - T

TEMPLATE_FONT_CONVERSION_FILE

Category Templates and Symbols

Points to the location of the Tekla Structures system font (Template Editor font) conversion file `template_fonts.cnv`. By default the path is `%DXK_FONTPATH%\template_fonts.cnv` meaning, for example, `C:\ProgramData\Trimble\Tekla Structures\<version>\ \environments\common\fonts\\template_fonts.cnv`. This file is used for mapping Tekla Structures system fonts (fixfont, romsim, romsim8, romco) to Windows fonts in DWG/DXF export.

The syntax: `<Template Editor font>= windows font [* width correction factor]`

For example: `romco = Times New Roman * 0.5`

There is another conversion file, `dxf_fonts.cnv`, that converts the True Type fonts to SHX fonts, for more information, see .

This advanced option is system specific and is read from environment files. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

XS_TEMPLATE_DIRECTORY

Category: File locations

Points to the folder containing template (`*.tpl`) and report (`*.rpt`) files.

This advanced option is model specific and the setting is saved in the options database.

You can use semicolon-separated lists of folder paths.

XS_TEMPLATE_DIRECTORY_SYSTEM

Category

This advanced option must be set in an initialization (.ini) file.

This advanced option is system specific and is read from environment files. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

This advanced option is used in `env_<environment_name>.ini` environment initialization files to define the location of environment-specific

templates (`.tpl`) and reports (`.rpt`). You can use semicolon-separated lists of folder paths.

Other users than administrators can define local folders using the advanced options `XS_FIRM`, `XS_PROJECT` and `XS_TEMPLATE_DIRECTORY`.

Example

```
set XS_TEMPLATE_DIRECTORY_SYSTEM=%XSDATADIR%\environments\uk
\general\template\
```

See also

XS_TEMPLATE_DIRECTORY (page 447)

XS_FIRM (page 260)

XS_PROJECT (page 360)

XS_TEMPLATE_MARK_SUB_DIRECTORY

Category

File Locations

Use this advanced option to change the name of the subfolder where Tekla Structures searches the templates that you use in marks. When you are adding a template in a mark, the available templates are displayed in the **Mark content - template** dialog box. You cannot define several folders by using semicolon-separated lists of folder paths.

By default, `mark` is the value for this advanced option. You can create another folder with another name and save your mark templates there, and enter the name of that folder as the value for this advanced option.

This advanced option is model specific and the setting is saved in the options database.

Example

Example of using another folder:

```
XS_TEMPLATE_MARK_SUB_DIRECTORY=my_mark_tpl
```

The mark templates are in this example case searched from the following folders in the following order:

```
%XS_TEMPLATE_DIRECTORY%\my_mark_tpl
```

```
ModelDir\my_mark_tpl
```

```
%XS_PROJECT%\my_mark_tpl
```

```
%XS_FIRM%\my_mark_tpl
```

`%XS_TEMPLATE_DIRECTORY_SYSTEM%\my_mark_tpl`

`%XS_SYSTEM%\my_mark_tpl`

See also

XS_TEXT_ORIENTATION_EPSILON

Category

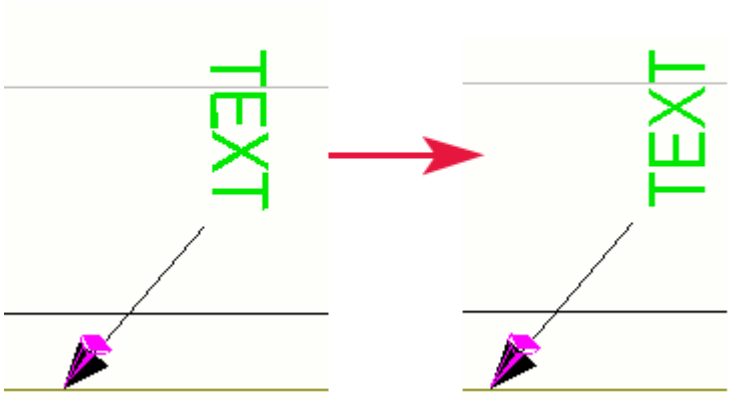
Drawing Properties

Use to define the point where text or dimension mark text that is positioned almost vertically is turned so that it is faced the other way around.

The default is `0.1`, which is 5.72958 degrees. For example, if you want the text to change (flip) at 100 degrees (or 10 off 90), you need to set this advanced option to to .175.

This advanced option is model specific and the setting is saved in the options database.

Example



See also

XS_THICKNESS_PARAMETER_IS_CROSS_SECTION_THICKNESS

Category

Profiles

Use to define the method to measure the thickness of parts (flanges, plates, walls and so on). When set to `FALSE`, the thickness parameter of the profile defines the actual thickness. When set to `TRUE`, the thickness parameter defines the cross section thickness (which is not same as the actual thickness if the part is sloped).

The default value is `FALSE`. We recommend that you use this value.

The change applies to the following profile types:

- SPD
- EPD
- I
- RHS
- PD
- P

XS_TPLED_INI

Category

File Locations

Defines the location of the `tpled.ini` file.

This is a system-specific advanced option.

Example

```
..\ProgramData\Trimble\Tekla Structures\<version>\
\environments\default\template\settings\
```

XS_TRY_TO_KEEP_LOCATION_IN_FREEPLACING

Category

Drawing Properties

Use to fine-tune the functionality of the **Arrange Drawing Objects** command in drawings.

This advanced option is set to `TRUE` by default, meaning that the **Arrange Drawing Objects** command tries to find a new location for the selected object as close to the current location as possible. If the current location is free, the object is not moved at all.

When this advanced option is set to `FALSE`, the **Arrange Drawing Objects** command works the same way as **Ignore Current Locations**.

This advanced option is model specific and the setting is saved in the options database.

XS_TUBE_UNWRAP_LIMIT_THICKNESS

Category

Drawing Properties

Use this advanced option to define how Tekla Structures draws tubes.

Set to a decimal value. Tekla Structures individually draws the inner and outer surfaces of tubes that are thicker than this value.

This advanced option is model specific and the setting is saved in the options database.

NOTE If the advanced option `XS_TUBE_UNWRAP_WITH_CUT_HOLES` is set to `TRUE`, this advanced option has no effect.

See also

XS_TUBE_UNWRAP_WITH_CUT_HOLES (page 452)

XS_TUBE_UNWRAP_PAPER_THICKNESS

Category

Drawing Properties

This advanced option lengthens unfolded tubes by the following multiplication factor:

factor = 1.0+2*`XS_TUBE_UNWRAP_PAPER_THICKNESS`/diameter

The default value is 0.0.

NOTE If the advanced option `XS_TUBE_UNWRAP_WITH_CUT_HOLES` is set to `TRUE`, this advanced option has no effect.

This advanced option is model specific and the setting is saved in the options database.

Example

Tube d=219

When you set the advanced option to 0, the unwrapped length of the tube (1.0*PI*diameter) = 688.

When the advanced option is set to 10, the length of the unfolded tube = factor * diameter* PI = (1.0 + 2*10/219) * 219 * 3.14 = 751

See also

XS_TUBE_UNWRAP_WITH_CUT_HOLES (page 452)

XS_TUBE_UNWRAP_USE_PLATE_PROFILE_TYPE_IN_NC

Category

CNC

To use the plate profile type B in the NC file header data for unwrapped round tubes, set this advanced option to `TRUE`. To use RO for round tubes, set it to `FALSE`. The default value is `TRUE`.

This advanced option only works for straight tubes, not for polybeam tubes.

This advanced option is model specific and the setting is saved in the options database.

WARNING Use this advanced option only when you have set the advanced option `XS_TUBE_UNWRAP_WITH_CUT_HOLES` to `TRUE`.

See also

XS_TUBE_UNWRAP_WITH_CUT_HOLES (page 452)

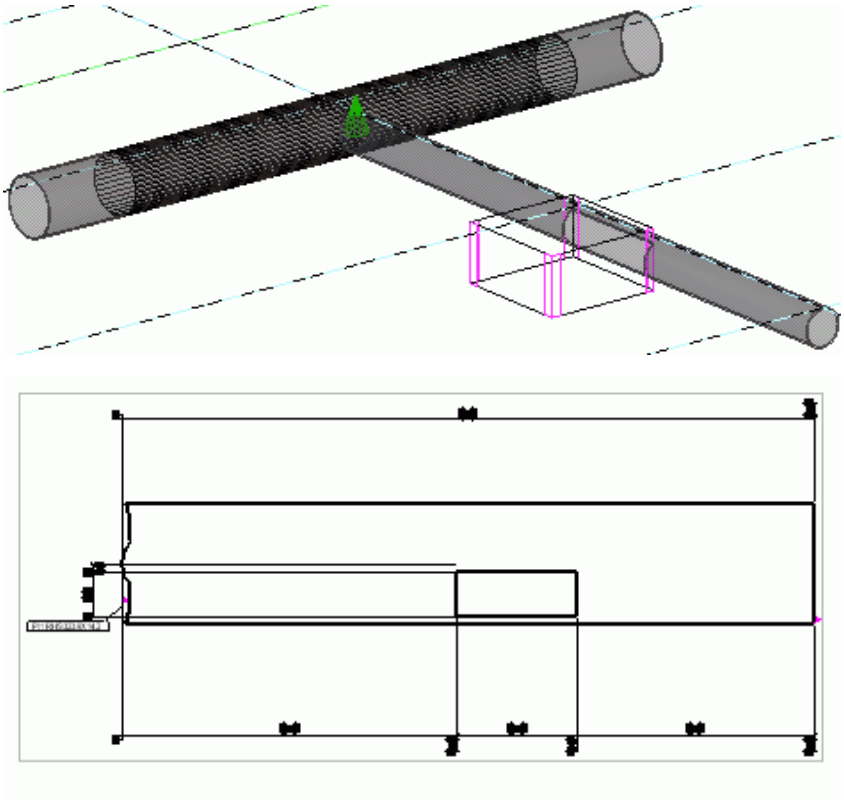
XS_TUBE_UNWRAP_WITH_CUT_HOLES

Category

Drawing Properties

Set to `TRUE` to include cut holes in workshop drawings or NC files of unwrapped CHS (circular hollow section) profiles.

This advanced option is set to `TRUE` by default. If you set it to `FALSE`, then the previously existing unwrapping method is applied.



This advanced option is model specific and the setting is saved in the options database.

NOTE If you set this advanced option to `TRUE`, the advanced options `XS_TUBE_UNWRAP_LIMIT_THICKNESS` and `XS_TUBE_UNWRAP_PAPER_THICKNESS` have no effect.

Note also that using this advanced option does not affect conical tube profiles.

See also

XS_TUBE_UNWRAP_LIMIT_THICKNESS (page 451)

XS_TUBE_UNWRAP_PAPER_THICKNESS (page 451)

XS_TUBE_UNWRAP_USE_PLATE_PROFILE_TYPE_IN_NC (page 452)

1.20 Advanced options - U

XS_UEL_IMPORT_FOLDER

This advanced option must be set in an initialization (.ini) file. It is system-specific.

You can collect all the `.uel` files exported from your custom components and sketched profiles in folders, and then automatically import them to new models. Use this advanced option to point to the folders which contain the `.uel` files. Note that you must still manually export the desired custom components and sketched profiles to these folders.

You can point to several folders if you separate the folders with a semicolon. For example:

```
set XS_UEL_IMPORT_FOLDER=%XSDATADIR%\environments\default
\components_sketches\;%XSDATADIR%\environments\common\components_sketches
\concrete\;%XSDATADIR%\environments\common\components_sketches\steel
\;%XSDATADIR%\environments\common\components_sketches\
```

When this advanced option is set (preferably in the `user.ini` file), and you create a new model, Tekla Structures automatically imports the `.uel` files to the model.

NOTE To take the custom components and related sketched profiles in use in existing models, import the `.uel` files through the **Applications & components** catalog. If the `.uel` files only contain sketched profiles, import them through the profile catalog.

XS_UNDERLINE_AFTER_POSITION_NUMBER_IN_HARDSTAMP

Category

CNC

Set the advanced option to `TRUE` to have an underscore (_) after the part position (without an assembly position) in a hardstamp.

The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_UNFOLDING_ANGLE_DIM_FORMAT

Category

Dimensioning: Unfolding

Use to define the format of angle text by entering an integer 0 - 7. The default value is 1. Enter one of the following values:

- 0 = ###
- 1 = ###[.#]
- 2 = ###.#
- 3 = ###[.##]
- 4 = ###.##
- 5 = ###[.###]
- 6 = ###.###
- 7 = ### #/#

This advanced option is model specific and the setting is saved in the options database.

XS_UNFOLDING_DONT_USE_NEUTRAL_AXIS_FOR_RADIUS

Category

Dimensioning: Unfolding

Set this advanced option to `FALSE` to use the neutral axis of the part to calculate the bending radius in unfolded views. Set this advanced option to `TRUE` to calculate the bending radius from the inner surface of the part. The default value is `TRUE`.. This advanced option only affects curved polybeams.

This advanced option is model specific and the setting is saved in the options database.

Note that this advanced option does not work if XS_USE_OLD_POLYBEAM_LENGTH_CALCULATION (page 477) is set to `TRUE`.

NOTE The settings in the `unfold_corner_ratios.inp` override this advanced option.

XS_UNFOLDING_ANGLE_DIM_PRECISION

Category

Dimensioning: unfolding

Use to define the precision of angle text. Enter an integer 1 - 10. The default value is 10. Enter one of the following values:

- 1 = 0.00

- 2 = 0.50
- 3 = 0.33
- 4 = 0.25
- 5 = 1/8
- 6 = 1/16
- 7 = 1/32
- 8 = 1/10
- 9 = 1/100
- 10 = 1/1000

Values 1 - 4 are intended for defining precision with rounding. For example, with precision 0.33 the actual dimension 50.40 is shown as 50.33. Values 5 - 7 are used for imperial units only. Values 8 - 10 are used for defining precision without rounding.

This advanced option is model specific and the setting is saved in the options database.

XS_UNFOLDING_PLANE_EPSILON

Category

Dimensioning: Unfolding

Use to specify the limit for checking if all section points are on the same plane. If the points are on the same plane, the part can be unfolded. Enter the value in millimeters.The default value is 0.01 mm.

This advanced option is model specific and the setting is saved in the options database.

XS_UNIQUE_NUMBERS

Category

Numbering

Set this advanced option to `TRUE` to have Tekla Structures create unique position numbers for all parts when numbering, even if they are equal. The default value is `FALSE`.

See also

XS_UNIQUE_ASSEMBLY_NUMBERS (page 457)

XS_UNIQUE_ASSEMBLY_NUMBERS

Category

Numbering

Set to `TRUE` if you want Tekla Structures to create unique position numbers for all assemblies when numbering, even if they are identical.

The default value is `FALSE`.

Parts are still numbered the same way as before.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_UNIQUE_NUMBERS (page 456)

XS_UPDATE_MARK_PLACING_IN_DRAWING

Category

Marking: Parts

Updates the position of updated marks in specific drawing types. Use the letters in the following table to specify the drawing types.

Letter	Drawing type
W	Single-part drawings
A	Assembly drawings
M	Multidrawings
G	General arrangement drawings
C	Cast unit drawings

The default value is AMW, which means that the mark position is updated in assembly drawings, multidrawings and single-part drawings.

This advanced option is model specific and the setting is saved in the options database.

Example

To update the position of updated part marks in single-part, assembly, multi-, and general arrangement drawings:

```
XS_UPDATE_MARK_PLACING_IN_DRAWING=WAMG
```

XS_UPDATE_MARKS_IN_FROZEN_DRAWINGS

Category

Marking: General

Set this advanced option to `TRUE` to automatically update marks in frozen drawings and create new marks if new parts have been added. If you set it to `FALSE`, Tekla Structures only updates parts and bolts. The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

XS_UPLOAD_SHARED_MODEL_TO_CONNECT

Category

Multi-user

Use this advanced option to select when a Tekla Structures model that has been shared with Tekla Model Sharing is uploaded to a folder in the attached Trimble Connect folder. The shared model will be uploaded to this folder as a read-only `.tekla` reference model that you can use in Tekla Structures and Trimble Connect.

The options are:

- `BASELINE` (default) = The shared model is automatically uploaded to the set Trimble Connect project folder each time a user creates a new baseline.
- `WRITEOUT` = The shared model is automatically uploaded to the set Trimble Connect project folder after each successful write out.
- `<NO VALUE>` = The shared model is never uploaded to the set Trimble Connect project folder.

Note that the path to the Trimble Connect project folder is set with the `XS_CONNECT_UPLOAD_MODEL_FOLDER (page 130)` advanced option.

This advanced option is model specific and the setting is saved in the options database.

XS_UPSIDE_DOWN_TEXT_ALLOWED

Category: Concrete Detailing

Set this advanced option to `TRUE` to have the text reading direction indicate the part installation direction. If there are similar asymmetric parts with

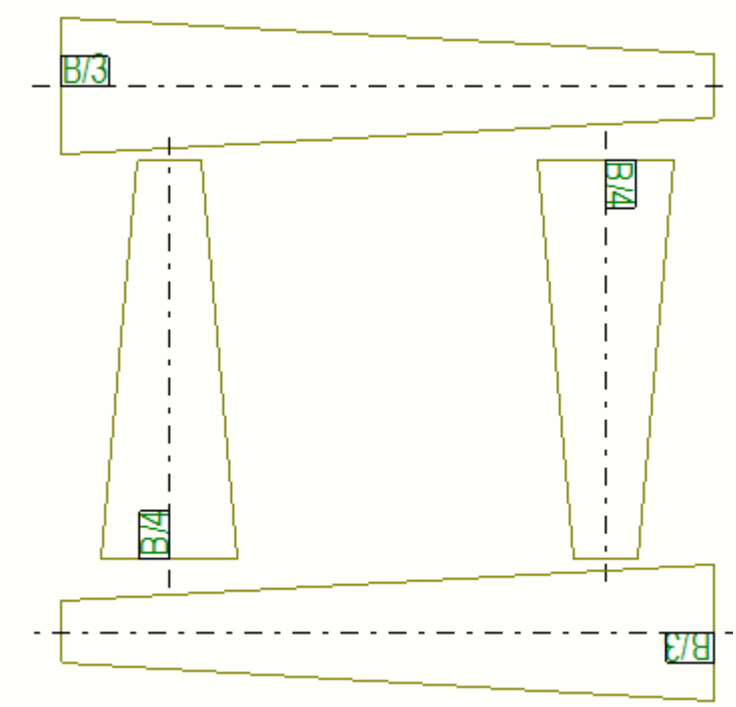
different orientation, some part marks may be placed upside down. When you set the advanced option to `FALSE` (default), none of the part marks are created upside down, so the text reading direction does not indicate the installation direction. If you set this advanced option to

- `CONCRETE`, only concrete marks and texts are upside down
- `STEEL`, only steel marks and texts are upside down
- `TIMBER`, only timber marks and texts are upside down

You can also use combinations of values separated by a comma, for example, `CONCRETE,STEEL`.

NOTE You need to use uppercase in the value, because writing `'true'` only affects free texts, whereas writing `'TRUE'` affects both free texts and part marks.

Example of the mark orientation when the advanced option is set to `TRUE`.



This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

For more information about indicating part orientation with marks, for example, see .

XS_USABSOLUTE_TO_RELATIVE_LIMIT

Category

Dimensioning: General

Use to affect the distance within which the first and last US Absolute dimension lines appear as relative. By default, this advanced option is not set to any value.

Example

```
XS_USABSOLUTE_TO_RELATIVE_LIMIT=1000
```

In this example, Tekla Structures will change first and last dimension lines shorter than 1000 mm to Relative dimensions. Dimension lines greater or equal to 1000 mm will remain US Absolute dimensions.

This advanced option is model specific and the setting is saved in the options database.

XS_USABSOLUTE2_TO_RELATIVE_LENGTH_FACTOR

Category

Dimensioning: General

Tekla Structures multiplies the space required by US Absolute2 dimension text by this value. If the result is larger than the actual dimension, Tekla Structures changes the dimension type to relative. The default value is `1.5`.

This advanced option is model specific and the setting is saved in the options database.

XS_USE_ANTI_ALIASING_IN_DX

Category: **Model View**

Use this advanced option to control whether antialiasing is used in the DirectX rendered model views. Antialiasing makes the edge lines smoother but with low resolution screens it can make the lines look thicker.

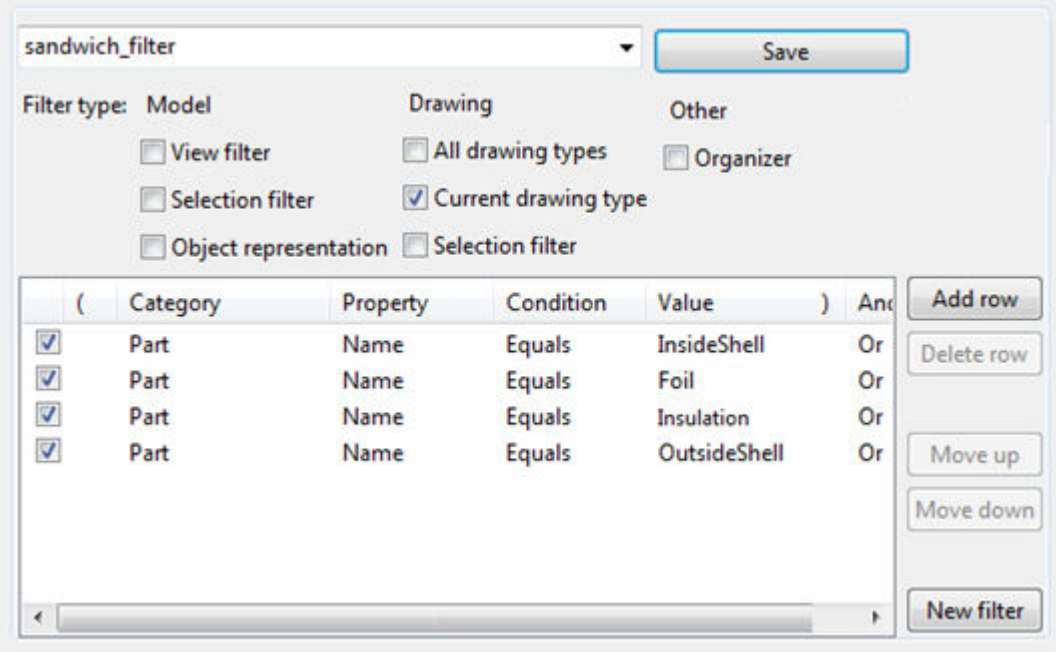
By default, this advanced option is set to `TRUE`.

XS_USE_ASSEMBLY_EXTREMA_IN_MARK_PLACING

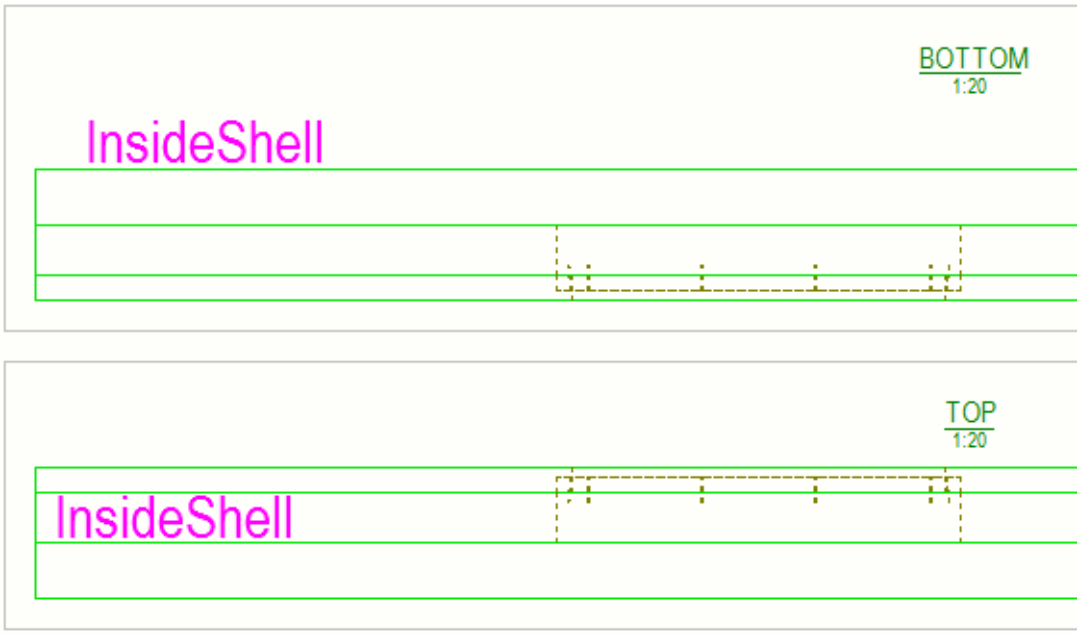
Category - Marking: General

Use this advanced option to place part marks on top of the assembly instead of on top of the main part. First create a drawing view filter and enter the name of the filter as the value. This advanced option can only be used for main parts of assemblies or cast units, not for secondary parts.

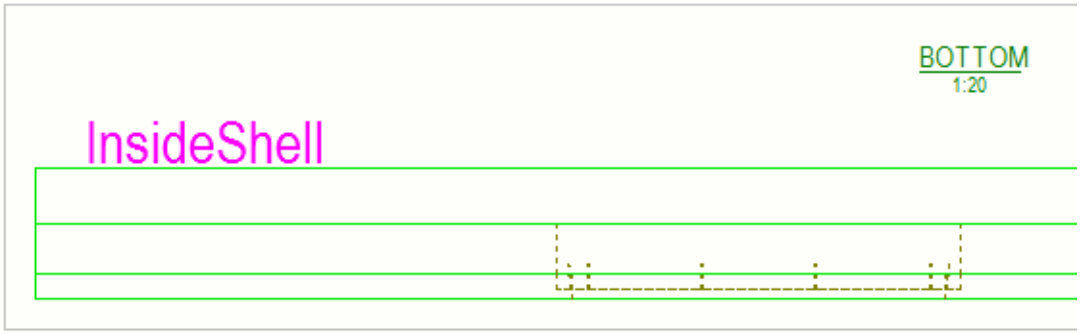
The example below uses part names as the filtering criteria.

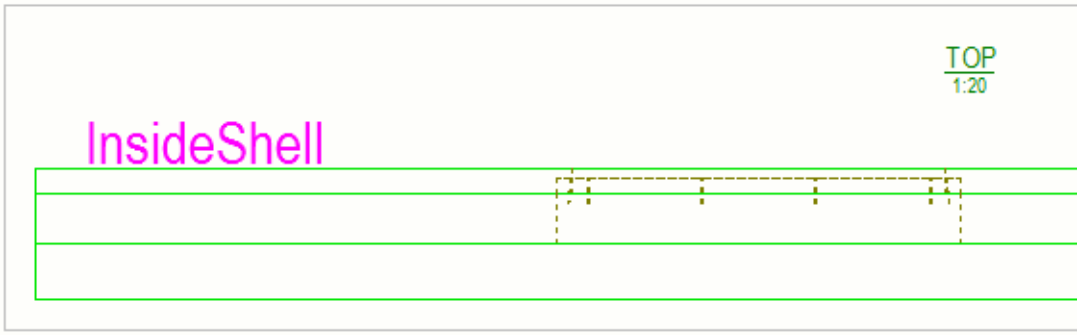


When you (re)create the drawing, the mark place changes from this:

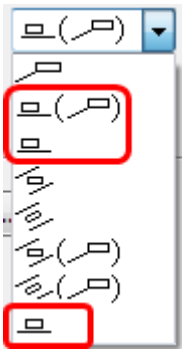


To this:





The supported mark placing types are shown below.



XS_USE_ASSEMBLY_NUMBER_FOR

Category: Numbering

Use this advanced option if you want the assembly/cast unit number to be also the main part number for the corresponding assembly/cast unit. Use one of the following options:

- Leave this option blank for each part in the assembly to get a part number, regardless of whether the assembly contains only one main part or several parts. Click the link to see an example of one part only or of multiple parts.

- Set to `MAIN_PART` to always assign the assembly or cast unit number to the main part of an assembly or cast unit. All other parts, if any, will use part number. Click the link to see an example of one part only or of multiple parts.
- Set to `LOOSE_PART` to assign the assembly or cast unit number to the main part of an assembly or cast unit that has no other parts. If the assembly or cast unit has several parts, the main part will receive a part number. Click the link to see an example of one part only or of multiple parts.

  The assembly prefix replaces the part prefix.

NOTE Do not use the same prefix for parts and assemblies.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_USE_BOLT_DISTANCE_IN_NOTCH_CALCULATIONS

This advanced option must be set in an initialization (.ini) file.

Set this advanced option to `TRUE` to use automatic notch height calculation according to bolt distance. This affects connections 129 and 184.

XS_USE_COLOR_DRAWINGS

Category

Drawing View

Changes the default color mode in drawings when the Tekla Structures is started. If you set this advanced option to `FALSE` or leave the value out, drawings are black and white. Set it to `GRAY` to have gray scale drawings. Set it to any other value, for example, `COLOR`, `TRUE` or `1`, to use colors in drawings. `TRUE` is the default value.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_USE_CONVEX_PROTECT_AREA

This advanced option must be set in an initialization (.ini) file.

Set this advanced option to `TRUE` to automatically calculate the protection area more accurately along the faces of parts, so that part marks can be placed inside the profile also for curved hollow sections. To protect the area of the object covers, enter `FALSE`. The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_USE_CROSS_FOR_OPENING_SYMBOL

Category

Drawing Properties

Use `XS_USE_CROSS_FOR_OPENING_SYMBOL` to select how the openings/recesses are shown and the symbols to be used.

The default value is `TRUE`, which means that a cross is used as the opening/recess symbol.

This advanced option is model specific and the setting is saved in the options database.

Value	Description
`TRUE` This is the default.	Crosses are used as symbols for openings as follows: If the recess is on the front face of the part, recess symbol and bounding lines are shown as unbroken lines as follows: If the recess is on the back face of the part, recess symbol and bounding lines are shown as dashed lines as follows:

Value	Description
FALSE	Shadows are used as symbols for openings as follows: If the recess is on the front face of the part, there is no recess symbol, and bounding lines are shown as unbroken lines as follows: If the recess is on the back face of the part, there is no recess symbol, and bounding lines are shown as dashed lines as follows:

XS_USE_DRAWING_NAME_AS_PLOT_FILE_NAME

Category

This advanced option must be set in an initialization (.ini) file.

Set this advanced option to `TRUE` to prevent Tekla Structures from converting the dot in the drawing name to an underscore in the plot file name when printing, for example B.1 to B_1. The default value is `FALSE`.

XS_USE_DYNAMIC_ROW_WIDTH_IN_TEMPLATES

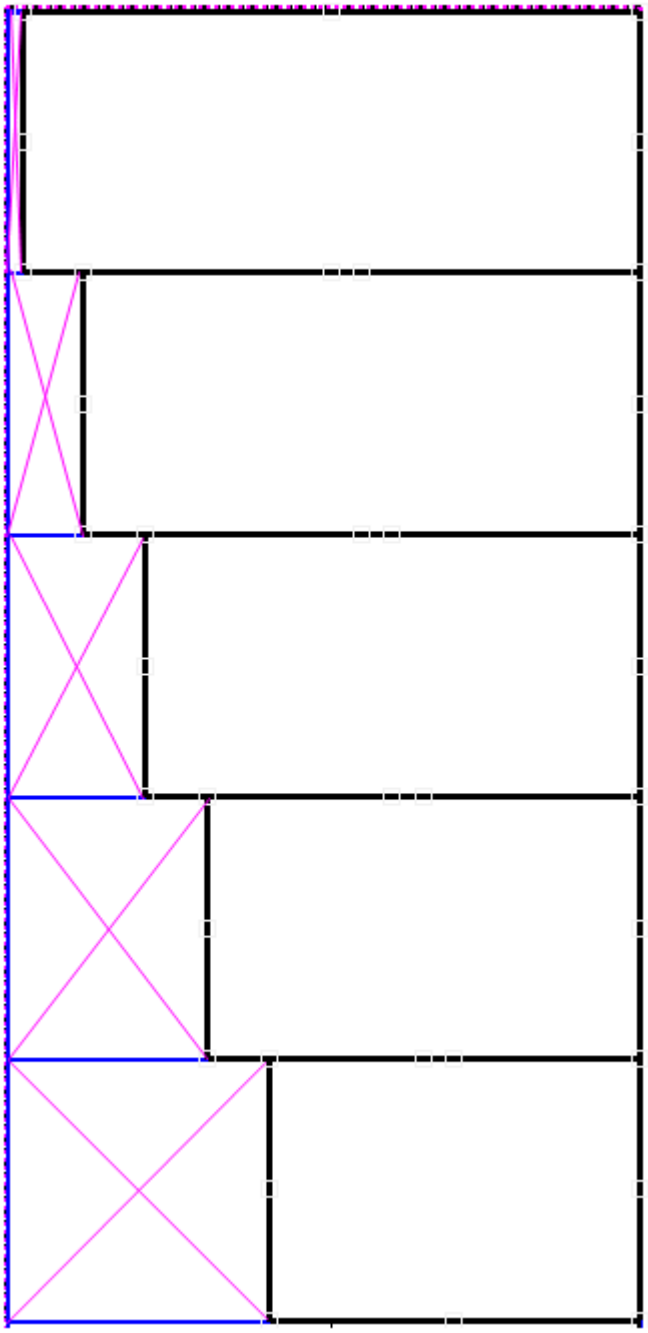
Category

This advanced option is available only in initialization files.

This advanced option does not work in report templates.

Set this advanced option to `TRUE` to fit the template row width dynamically according to the content, for example, drawing frames according to the different drawing sizes. The content must always be located on the right. If you set this advanced option to `FALSE`, automating fitting of the templates rows is not in use.

The default value is `FALSE`.



NOTE This functionality is not available in drawing part mark templates.

XS_USE_DRAWING_NAME_AS_PLOT_TITLE

Category

Printing

Set this advanced option to `TRUE` when you want to use the drawing name as the print title, for example, when printing to a `.pdf` file or to a Windows printer. To use the general Tekla Structures print title, such as "Tekla Structures drawing - A [T.100]", set it to `FALSE`. The default value is `TRUE`.

As a result, the Windows printer dialog box, and the pdf file name will contain the drawing print file name you define with the advanced options listed below.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_DRAWING_PLOT_FILE_NAME_A (page 211)

XS_DRAWING_PLOT_FILE_NAME_C (page 215)

XS_DRAWING_PLOT_FILE_NAME_W (page 212)

XS_DRAWING_PLOT_FILE_NAME_G (page 213)

XS_DRAWING_PLOT_FILE_NAME_M (page 214)

XS_USE_EIGHT_COLORS_IN_MODELING_VIEWS

Set this advanced option in the `user.ini` file located in `..\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`.

Set to `TRUE` to disable additional colors in model views. The default value is `FALSE`.

XS_USE_EXACT_SOLID_FOR_CLASH_CHECK

Category

Speed and Accuracy

If you set this advanced option to FALSE (default), normal solid accuracy is used in clash checking. If you need to use high solid accuracy in clash checking, set this advanced option to TRUE. This advanced option is model specific.

WARNING Using high accuracy, that is, setting this advanced option to TRUE, slows down the clash checking process, and there is a higher risk of solid errors.

XS_USE_EXISTING_SINGLE_PART_DRAWINGS_IN_ASSEMBLY_DRAWINGS

Category

Single Part View in Assembly Drawing

You can specify whether to create new views or to use views from existing single part drawings in assembly drawings. When this advanced option is set to TRUE, existing single part drawing views will be used in assembly drawings. When set to FALSE, or if there is no existing single part drawing for a given part, a new view will be created according to the **Single-part attributes** setting (**Assembly drawing properties** --> **Layout** --> **Other**). The default value is FALSE.

NOTE This setting only works with assembly drawings, not with multidrawings

This advanced option is model specific and the setting is saved in the options database.

XS_USE_EXISTING_SINGLE_PART_DRAWINGS_SCALE

Category: Single Part View in Assembly Drawing

If you do not want to maintain the scale of the existing single part drawing included in an assembly drawing, set the advanced option XS_USE_EXISTING_SINGLE_PART_DRAWINGS_SCALE to FALSE. When you do this, the scale of the included single part drawing will follow the scale of the assembly drawing, or advanced option XS_SINGLE_SCALE (page 429) if it is set.

Tekla Structures maintains the original scale in a single part drawing in an assembly drawing when you set the layout to include single part drawings, and set the option

XS_USE_EXISTING_SINGLE_PART_DRAWINGS_IN_ASSEMBLY_DRAWINGS (page 468) is set to TRUE.

This advanced option is model specific and the setting is saved in the options database.

XS_USE_FLAT_DESIGNATION

Category

Plate Work

Set this advanced option to TRUE to use flat bar designation, which means that Tekla Structures shows plates as the equivalent flat bars for manufacturing. The default value is FALSE.

Indicate the prefix you want to use for flat bars using the advanced option XS_FLAT_PREFIX (page 261).

This advanced option is model specific and the setting is saved in the options database.

See also

XS_USE_INTEGRATED_BUILDING_HIERARCHIES

Category: Modeling Properties

To define building levels and hierarchies in the **Building hierarchy** dialog box, set this advanced option to TRUE.

The default value is FALSE.

This advanced option is model specific and the setting is saved in the options database.

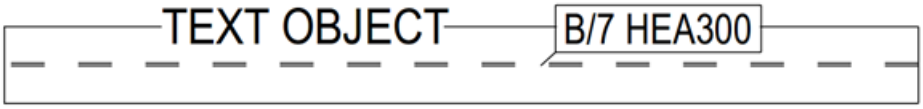
Reopen the model after changing the value to activate the new setting.

XS_USE_LINECLIP

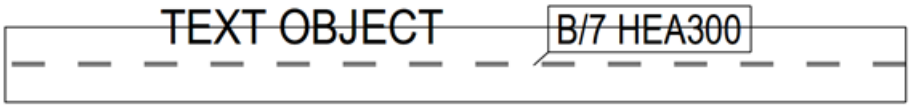
Category: Printing

Set this advanced option to TRUE to clip continuous lines at objects in drawing printouts (paper or .pdf). Set to FALSE to display continuous lines, for example, to run the line through text or drawing marks. The default value is TRUE.

`XS_USE_LINECLIP` is set to `TRUE`:



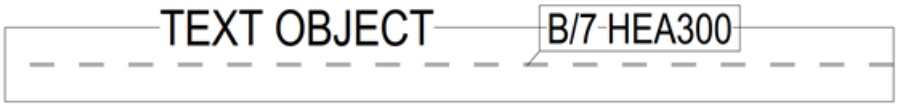
`XS_USE_LINECLIP` is set to `FALSE`:



Note that this setting only affects drawings printed using the old printing functionality (`XS_USE_OLD_PLOT_DIALOG=TRUE`), and it does not completely match what you see in drawings in Tekla Structures. If `XS_USE_OLD_PLOT_DIALOG` is set to `FALSE` (default), then `XS_USE_LINECLIP` has currently no effect, and the lines are clipped in the same way in drawings and in printouts.

This is how the line clipping is displayed in drawings currently:

This is how it is printed:



This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

XS_USE_LONG_POINTS_IN_DIMENSIONING

Category

Dimensioning: Parts

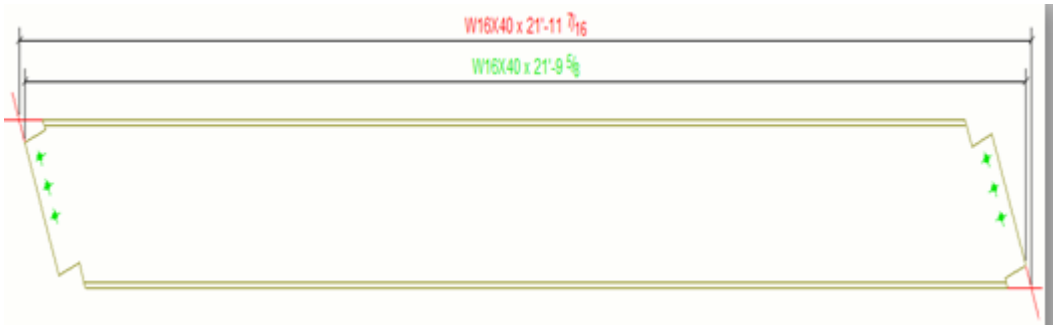
Sometimes there may be a need to dimension the parts in assembly or single-part drawings so that copes or notches are not taken into account, for example, to estimate the space needed for transportation.

If you set this advanced option to `TRUE`, the overall dimensions are calculated to long points. If you set it to `FALSE`, the overall dimensions are calculated to cope points. `FALSE` is the default.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

Example

In the example below, the upper dimension (the red one) shows the result when this advanced option is set to `TRUE`, and the lower one when it is set to `FALSE` (the green one).



WARNING Using this advanced option will not affect the length of the part in the BOM, reports, or CNC.

XS_USE_MODEL_PREFIX_IN_MULTI_NUMBERS_FOR

Category: Numbering

Use to allow the prefixes used in part and assembly numbering to be used in multidrawing numbers. Enter any of the following options: `NONE`, `ASSEMBLIES`, `PARTS`, and `ASSEMBLIES_AND_PARTS`. The default value is `ASSEMBLIES_AND_PARTS`.

This advanced option is model specific and the setting is saved in the options database.

Example

If you set this advanced option to `PARTS`, the multinumbers for parts appear as `101Pa`.

See also

XS_USE_MULTI_NUMBERING_FOR (page 472)

XS_MODEL_PREFIX_INFLUENCES_MULTI_NUMBERING_FOR (page 322)

XS_USE_MULTI_NUMBERING_FOR

Category: Numbering

Use to define if multinumbering affects assemblies, parts, or both. Part and assembly numbering must be based on drawing numbers to use multinumbering.

The options are:

- `NONE`: No assemblies or parts will get multi-numbered, even if linked to multidrawings.
- `ASSEMBLIES`: Assemblies will get multinumbered, but parts will not. This is the default US Imperial Steel setting.
- `PARTS`: Only parts will get multinumbered. Common if you are creating drawings for assemblies one per sheet, and handling parts on large collection sheets grouped by plates, or angles, for example.
- `ASSEMBLIES_AND_PARTS`: Both assemblies and parts will get multinumbering, but how is determined by workflow and other settings.

The default value is `ASSEMBLIES_AND_PARTS`.

WARNING	Do not change the value during a project.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_MULTI_NUMBERING_INCLUDE_ASSEMBLY_PARTS (page 324)

XS_USE_MULTI_NUMBERING_WHEN_COPYING_DRAWING_VIEWS (page 473)

XS_USE_NUMERIC_MULTI_NUMBERS_FOR (page 475)

XS_MODEL_PREFIX_INFLUENCES_MULTI_NUMBERING_FOR (page 322)

XS_USE_MODEL_PREFIX_IN_MULTI_NUMBERS_FOR (page 471)

XS_SWITCH_MULTI_NUMBERS_FOR (page 444)

XS_PART_MULTI_NUMBER_FORMAT_STRING (page 340)

XS_ASSEMBLY_MULTI_NUMBER_FORMAT_STRING (page 85)

XS_CAST_UNIT_MULTI_NUMBER_FORMAT_STRING (page 109)

XS_VALID_CHARS_FOR_PART_MULTI_NUMBERS (page 492)

XS_VALID_CHARS_FOR_ASSEMBLY_MULTI_NUMBERS (page 491)

XS_MIN_NUMBER_OF_PART_MULTI_CHARACTERS (page 319)

XS_MIN_NUMBER_OF_ASSEMBLY_MULTI_CHARACTERS (page 318)

XS_USE_MULTI_NUMBERING_WHEN_COPYING_DRAWING_VIEWS

Category: Numbering

Set this advanced option to `TRUE` to use multinumbering when copying drawing views. If you do not want to use multinumbering, set it to `FALSE`. The default value is `FALSE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_USE_MULTI_NUMBERING_FOR (page 472)

XS_USE_NEW_PLATE_DESIGNATION

Category

Plate Work

Use this advanced option to control if the width and length are switched in parts if the width is greater than the length. The options are:

- Option is not used: set value to blank or `FALSE`.
- Option is used only for steel parts: set value to `FOR_STEEL_PARTS_ONLY`.
- Option is used for all parts: set value to `TRUE`. This option is also used if value is set to anything else that does not match the other options above.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

Example

A beam has the profile BL15*240 and the distance between beam endpoints is changed to 215 mm:

- If `XS_USE_NEW_PLATE_DESIGNATION` is used, the length of the beam is fixed at 240, and the beam profile changes to BL15*215.

- If XS_USE_NEW_PLATE_DESIGNATION is not used, the length of the beam is 215 and the profile remains BL15*240.

XS_USE_NEW_WELD_PLACING

Category

Welds

If you have set welds visible in the drawing, this advanced option affects in which drawing view (front, back, top, or bottom) Tekla Structures draws the welds.

- When the advanced option is set to TRUE, Tekla Structures draws welds in the view that has the best visibility to the **secondary part** (default).
- When the advanced option is set to FALSE, Tekla Structures selects the view according to the **main part**.

This advanced option is model specific and the setting is saved in the options database.

See also

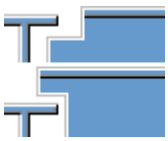
XS_USE_NEW_USNOTCH

Category

Components

Use to indicate whether to locate the horizontal cut of a notch above or below the flange of the main beam. The default value is TRUE. If you do not want to use the US style notch, set this advanced options to FALSE.

Used with the following notching options:



This advanced option is model specific and the setting is saved in the options database.

XS_USE_NUMBER_SELECTED_FOR_DRAWING_CREATION_AND_UPDATE

Category: Numbering

If the numbering is not up to date when you create a drawing, Tekla Structures asks you to number the model.

Set this advanced option to `TRUE` to number only the assemblies and parts that have the same numbering series as the selected part (or the main part of the selected drawing).

When set to `TRUE`, this advanced option does the same as if you clicked **Drawings & reports** --> **Number series of selected objects** . `TRUE` is the default value.

If you set this advanced option to `FALSE`, Tekla Structures numbers the whole model, which is the same as clicking **Drawings & reports** --> **Numbering** --> **Number modified objects** .

This advanced option is model specific and the setting is saved in the options database.

XS_USE_NUMERIC_MULTI_NUMBERS_FOR

Category: Numbering

Use to define which objects have numeric multi-numbers. The options are:

- `ASSEMBLIES`
- `PARTS`
- `ASSEMBLIES_AND_PARTS`
- `NONE`

This advanced option is model specific and the setting is saved in the options database.

Example

If you set this advanced option to `PARTS`, Tekla Structures displays the part multinumber as e.g. 101/1, instead of 101/a.

See also

XS_USE_MULTI_NUMBERING_FOR (page 472)

XS_USE_OLD_DRAWING_CREATION_SETTINGS

This advanced option must be set in an initialization (.ini) file.

Set the advanced option `XS_USE_OLD_DRAWING_CREATION_SETTINGS` to `TRUE` to use old drawing functionality and old drawing view property dialog boxes and subdialog boxes. In this old approach, drawing object properties can be defined on both drawing and view level, not individually for each view, like in the new view-level approach. View-level dimensioning rules are not supported.

By default, this advanced option is not in use.

Where to change the value depends on your company or project size, and on which level you need to unify certain enterprise-level settings. You can set this advanced option to `TRUE` in the `options.ini` file under the current model folder, in company's own `company.ini` file, your company's own `role.ini` file, or the `options.ini` file in firm or project folders , for example.

Note that Trimble Solutions no longer maintains the old functionality and dialog boxes. In practice, this means that new features, like custom presentations or automatic view-level properties will not be available in the old dialog boxes.

XS_USE_OLD_DRAWING_EXPORT

This advanced option must be set in an initialization (.ini) file.

If you want to use the old DWG/DXF export, set the advanced option `XS_USE_OLD_DRAWING_EXPORT` to `TRUE` in an `.ini` file. This advanced option is by default set to `FALSE` .

For instructions on using the old export, see .

XS_USE_OLD_DRAWING_LIST_DIALOG

Category: Drawing Properties

Document manager is enabled by default instead of **Drawing list**. All commands and buttons that would have launched **Drawing list** in earlier Tekla Structures versions will from version 2018i onwards launch **Document manager**. If you want to enable the old **Drawing list** instead, set this advanced option to `TRUE`. If you do this, **Document manager** will be disabled.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`.

For details about **Document manager**, see .

For details about **Drawing list**, see .

XS_USE_OLD_PLOT_DIALOG

Category

Printing

Set this advanced option to `TRUE` to use **Printer Catalog** and Tekla Structures own printer instances in printing.

The default value is `FALSE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_USE_OLD_POLYBEAM_LENGTH_CALCULATION

Category

Dimensioning: Unfolding

Set this advanced option to `TRUE` to calculate polybeam length using the legacy method where lengths of straight parts are added together, without taking the unfolding into account. If you set it to `FALSE` (default) and use the new method, the polybeam length is defined by unfolding the polybeam first and then calculating the length. This way of calculating gives a more accurate value for the polybeam length.

This advanced option is model specific and the setting is saved in the options database.

NOTE Using this advanced option is not recommended, because the length may not be reported correctly in all cases, especially for polybeams with curved chamfers.

NOTE When you switch on this advanced option, other ways to calculate polybeam length are not used by Tekla Structures, for example, `XS_CALCULATE_POLYBEAM_LENGTH_ALONG_REFERENCE_LINE`, `XS_DONT_USE_NEUTRAL_AXIS_FOR_RADIUS` or the unfold parameter settings in file `unfold_corner_ratios.inp`.

See also

XS_CALCULATE_POLYBEAM_LENGTH_ALONG_REFERENCE_LINE (page 106)

XS_USE_ONLY_INCHES_IN_SHEET_SIZES

Category

Imperial Units

Set this advanced option to `TRUE` to have sheet sizes in layouts and drawing lists in inches. To have the sheet sizes in feet and inches, set it to `FALSE` (default).

In order for this advanced option to work, set the advanced options `XS_IMPERIAL` and `XS_IMPERIAL_INPUT` to `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

XS_USE_ONLY_INCHES_IN_WELD_LENGTH

Category

Imperial units

Set this advanced option to `TRUE` to only display inches in weld length symbols. If you do not want to do this, set this advanced option to `FALSE`. This advanced option only works when the imperial units are in use. The default value is `TRUE`.

When you only display inches means that instead of showing 1ft 2inch it shows 14inch, for example.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_USE_ONLY_NOMINAL_REBAR_DIAMETER

Category

Concrete Detailing

Nominal diameter is the diameter used for calculating the cross section area of the reinforcing bar. Actual diameter takes into account the ribs, and tells the smallest hole diameter where the bar fits.

Values used for nominal and actual diameter are defined in `rebar_database.inp`, which is located in the environment folders in `\<environment>\profil`.

Set the advanced option to `TRUE` to use nominal diameter. To use the actual diameter set this advanced option to `FALSE`. The default value is `FALSE`.

When the advanced option is set to `FALSE`, and you open a model created earlier than Tekla Structures version 18, the center line of the reinforcing bars stays in place and the concrete cover is reduced. All bending dimensions of the reinforcing bar increase. To solve this problem, either set the advanced option to `TRUE` or modify the concrete covers of all reinforcing bars to the correct value.

When reinforcing bars are exported to Unitechnik, you can select to export either nominal or actual diameters. For other exports (for example, BVBS), the nominal diameter is always used in the exported definitions regardless of this advanced option.

WARNING Do not change this option during a project.

Changing the advanced option also changes the modeled reinforcing bars. This means that if the actual diameter will be used, the reinforcing bar looks thicker in the model. To accommodate for the thicker reinforcing bar, Tekla Structures will automatically change also concrete cover thickness. When you change the option, Tekla Structures changes the concrete cover values after next restart.

This advanced option is role specific. When the type **SYSTEM(ROLE)** is in use, the default value is used. When the type **MODEL(ROLE)** or **DRAWING(ROLE)** is in use, you can change the value, which is then the same for all users in the current model.

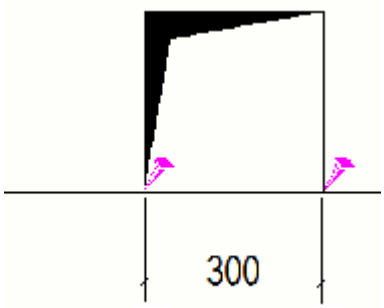
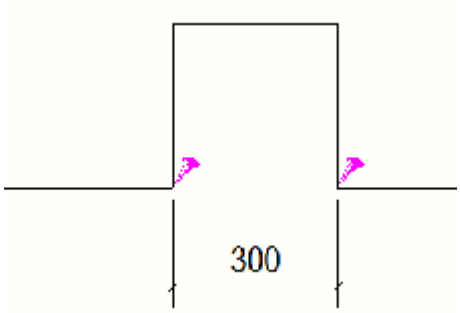
XS_USE_OPENING_SYMBOL_IN_BORDER_HOLES

Category

Drawing Properties

Use `XS_USE_OPENING_SYMBOL_IN_BORDER_HOLES` to select whether to use an opening symbol in openings located at part borders.

Value	Description
`TRUE`	Opening symbol is used in the openings located at the border of the part. The symbol used depends on the setting of the advanced option

Value	Description
	XS_USE_CROSS_FOR_OPENING_SYMBOL 
FALSE This is the default.	No opening symbol is used for openings at the border of the part. 

This advanced option is model specific and the setting is saved in the options database.

See also

XS_USE_CROSS_FOR_OPENING_SYMBOL (page 464)

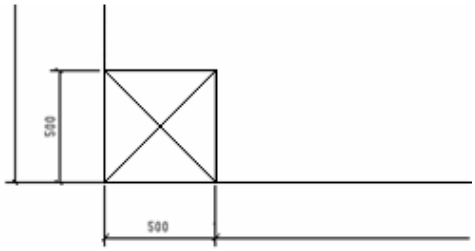
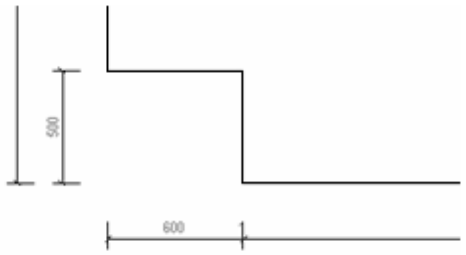
XS_USE_OPENING_SYMBOL_IN_CORNER_HOLES

Category

Drawing Properties

Use XS_USE_OPENING_SYMBOL_IN_CORNER_HOLES to select whether to use the opening symbol in openings located in part corners.

Value	Description
TRUE	Opening symbol is used in the openings located in the corner of the part. The symbol used

Value	Description
	depends on the setting of the advanced option XS_USE_CROSS_FOR_OPENING_SYMBOL. 
FALSE This is the default.	No opening symbol is used for openings in the corner of the part. 

This advanced option is model specific and the setting is saved in the options database.

See also

XS_USE_CROSS_FOR_OPENING_SYMBOL (page 464)

XS_USE_PLATE_SIDE_POSITIONING

Category

Dimensioning: Parts

Set this advanced option to TRUE to make the position dimension of plates dependent on the position of the plates in the model. When a plate is positioned underneath the work plane, Tekla Structures will place the position dimension on the top face of the plate. The default value is FALSE.

This advanced option is model specific and the setting is saved in the options database.

XS_USE_POINT_AS_SEPARATOR_IN_PROFILE_NAME

Category

Profiles

Set this advanced option to `TRUE` to use the period character (.) as the separator in parametric profile names, instead of using it as a decimal separator. This increases the number of separators available in the US imperial environment. The default value is `FALSE`.

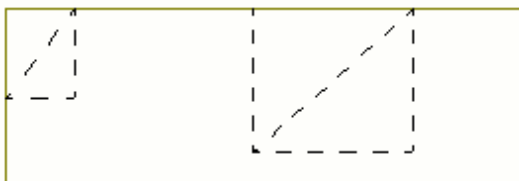
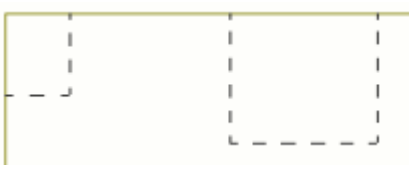
This advanced option is model specific and the setting is saved in the options database.

XS_USE_RECESS_SYMBOL_FOR_BORDER_AND_CORNER_RECESSES

Category: Drawing properties

Use `XS_USE_RECESS_SYMBOL_FOR_BORDER_AND_CORNER_RECESSES` to select whether to use the recess symbol in corner and border recesses. `TRUE` is the default value.

This advanced option is model specific and the setting is saved in the options database.

Value	Description
`TRUE`	Recess symbol is used in the recesses located at the border or in the corner of the part. The symbol used depends on the setting of the advanced option `XS_USE_CROSS_FOR_OPENING_SYMBOL` 
`FALSE`	No recess symbol is used for recesses at the borders or in the corners of the part. 

See also

XS_USE_CROSS_FOR_OPENING_SYMBOL (page 464)

XS_USE_OPENING_SYMBOL_IN_BORDER_HOLES (page 479)

XS_USE_OPENING_SYMBOL_IN_CORNER_HOLES (page 480)

XS_USE_REPAIR_NUMBERING_INSTEAD_OF_NUMBERING

Category: Numbering

If this advanced option is set to `TRUE`, Tekla Structures automatically repairs numbering instead of only running numbering.

When this advanced option is set to `TRUE`:

- Using the **Number modified objects** command does the same as using the **Diagnose and repair numbering: All** command
- Using the **Number series of selected objects** command does the same as using the **Diagnose and repair numbering: Series of selected objects** command

The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_USE_ROUND_MAIN_PART_COORDINATES_FOR_SECONDARY_PART_ANGLE

Category

Dimensioning: Parts

Set to `TRUE` to have the secondary part skewed dimensions and angle dimension use one of the main part directions if the main part profile is round or round tube. The default value is `TRUE`.

This advanced option is model specific and the setting is saved in the options database.

XS_USE_SCREW_POINT_ELEVATION_DIM

Category

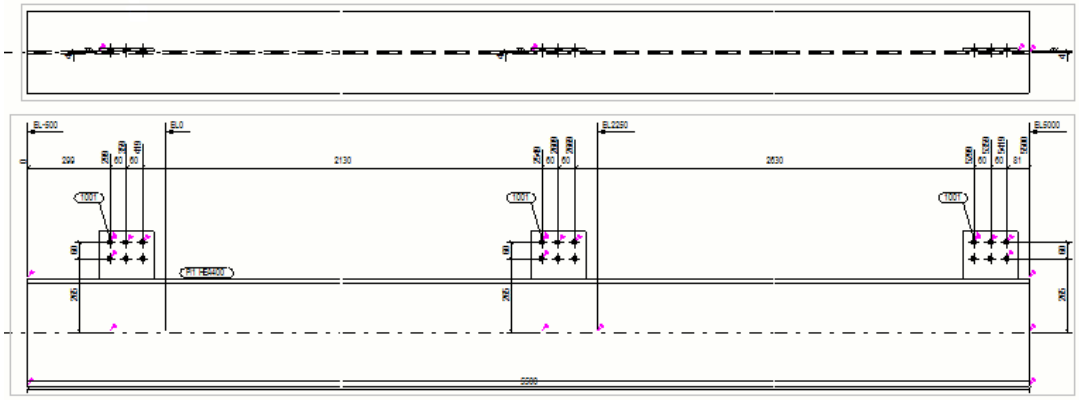
Dimensioning: Bolts

Set this advanced option to `TRUE` to display the elevation dimensions of a column to the working points of a neighboring part. If you set it to FALSE, the elevation dimensions are displayed at the ends of the column. The default value is `FALSE`.

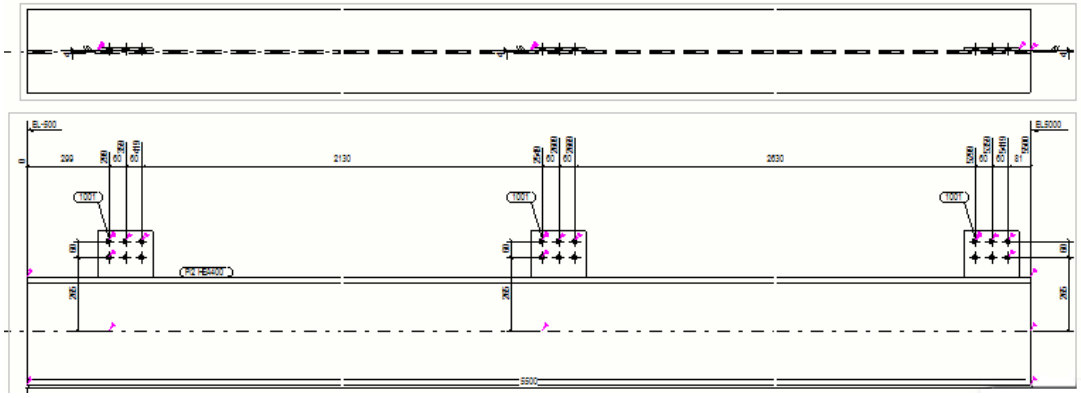
This advanced option is model specific and the setting is saved in the options database.

Example

An example, where the value is `TRUE`:



An example, where the value is `FALSE`:



XS_USE_SMALLER_GUSSET_PLATE

Category

Components

Set this advanced option to `TRUE` to minimize the size of rectangular gusset plates created by gusset connections. You can create smaller gusset plates by using a single bracing and secondary bolts dimensioned to the middle of the

secondary part. Tekla Structures generates a triangular gusset plate when the main part is located between diagonals. The default value is FALSE.

This advanced option is model specific and the setting is saved in the options database.

XS_USE_SMART_PAN

Category: Drawing View

Set this advanced option to TRUE to enable optimized zooming and panning in drawings. When you pan or zoom a heavy drawing with lots of graphics, and the smart pan is enabled, the whole screen is not drawn, and there is empty space on the edges, and zooming and panning is faster.

The whole screen is drawn again when you stop panning or zooming.

This advanced option is by default set to FALSE, because using optimization sometimes results in unwanted checkerboard effect.

This advanced option is user specific and the setting is saved in options.bin under the user folder, for example, in C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings. Restart Tekla Structures to activate the new value.

Restart Tekla Structures after changing the value to activate the new setting.

XS_USE_SMOOTH_LINES

Category

Model View

Set this advanced option to TRUE to use anti-aliasing to minimize jagged edges in model views. Before using this advanced option, check that your display adapter supports anti-aliasing. This advanced option is supported only for OpenGL rendering.

The default value is FALSE.

This advanced option is user specific and the setting is saved in options.bin under the user folder, for example, in C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings. Restart Tekla Structures to activate the new value.

XS_USE_SOFTWARE_RENDERING

Category

Model view

Set this advanced option to `TRUE` to bypass your graphic adapter in model views. Use this advanced option if you have problems with your display (for example, lines are not drawn correctly). The default value is `FALSE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_USE_SPECIAL_FILLER_PLATE_THICKNESS

Category

Profiles

Set this advanced option to `TRUE` to have shim plate thicknesses comply with Japanese standards. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_USE_TUBE_INNER_LENGTH_IN_DIMENSIONING

Category

Dimensioning: Parts

Set to `TRUE` to dimension the overall length of tube profiles along the inner surface instead of outer surface. The default value is `FALSE`.

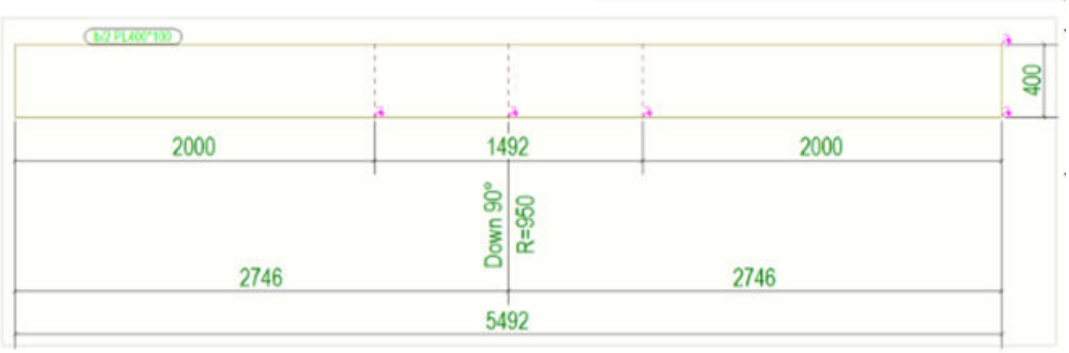
This advanced option is model specific and the setting is saved in the options database.

XS_USE_UP_DOWN_SIGN_INDICATOR_FOR_ANGLE_IN_UNFOLDING

Category: Dimensioning: Unfolding

Set this advanced option to `TRUE` to show Up and Down text instead of positive and negative angle values in unfolded single-part drawings for angle dimensions. `FALSE` is the default value.

Setting this advanced option to TRUE will omit the text specified for the advanced option XS_ANGLE_TEXT_IN_UNFOLDING_BENDING_LINE_DIMENSIONING.



This advanced option is model specific and the setting is saved in the options database.

See also

XS_DRAW_BENDING_END_LINE_DIMENSIONS_IN_UNFOLDING (page 180)

XS_DRAW_BENDING_END_LINES_IN_UNFOLDING (page 181)

XS_USE_USABSOLUTE_ARROW_TYPE_FOR_ABSOLUTE_DIMENSIONS

Category

Dimensioning: General

Set this advanced option to TRUE to use the arrow shape US Absolute also for normal absolute dimensions. FALSE is the default value.

You can select the arrow shape from the **US Absolute dimensions** list on the **Appearance** tab in the dimension properties dialog box.

Note that this advanced option only applies to GA drawing dimensions, and old dimensioning (XS_USE_OLD_DRAWING_CREATION_SETTINGS (page 475) is set to TRUE).

This advanced option is model specific and the setting is saved in the options database.

XS_USE_USER_DEFINED_REBAR_LENGTH_AND_WEIGHT

Category

Concrete Detailing

Set this advanced option to TRUE to calculate the length and weight of the reinforcing bars in **Rebar Shape Manager** using formulas in the fields L and WEIGHT.

If you set this advanced option to FALSE, the length and the weight are automatically calculated according to the center line of the reinforcing bars. The default value is FALSE.

To read the length and weight from **Rebar Shape Manager**, you also have to set XS_USE_USER_DEFINED_REBARSHAPERULES to TRUE.

NOTE This setting only affects reports. If you set this advanced option to TRUE and you have not defined the formulas for length and weight in **Rebar Shape Manager,** the values in the reports show zero (0).

This advanced option is model specific and the setting is saved in the options database.

See also

XS_USE_USER_DEFINED_REBARSHAPERULES (page 488)

XS_USE_USER_DEFINED_REBARSHAPERULES

Category

Concrete Detailing

Use this advanced option to define whether reinforcing bar bending shapes are recognized according to the bending shape definitions created with **Rebar Shape Manager** and saved in the RebarShapeRules.xml file.

This advanced option is set to TRUE by default, meaning that the bending shape recognition uses the reinforcing bar shapes saved in the RebarShapeRules.xml file.

If you set this advanced option to FALSE, the **Rebar Shape Manager** definitions are not used, and the definitions in rebar_schedule_config.inp are used instead. We recommend that you set this advanced option to TRUE and use **Rebar Shape Manager**.

This advanced option is model specific and the setting is saved in the options database.

XS_USE_VERTICAL_PLACING_FOR_COLUMNS_IN

Category: Drawing Properties

Use to place columns vertically in single part, assembly and cast unit drawings. Use the following options to specify the types of drawings where the columns should be placed vertically:

- `ASSEMBLY_DRAWINGS` - in assembly drawings and cast unit drawings only
- `SINGLE_PART_DRAWINGS` - in single part drawings only
- `ASSEMBLY_AND_SINGLE_PART_DRAWINGS` - in single part, assembly and cast unit drawings

This advanced option is model specific and the setting is saved in the options database.

XSUSERDATADIR

Category

This advanced option must be set in an initialization (.ini) file.

This advanced option is system specific and is read from `teklastructures.ini`. Generally, there is no need to modify system-specific settings. Do not modify them if you are not an administrator.

Example

`set XSUSERDATADIR=%LOCALAPPDATA%\Tekla Structures\<version number>`.

For example, `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`.

XS_USER_DEFINED_BOLT_SYMBOL_TABLE

Category

Marking: Bolts

Defines the location of the user-defined bolt symbol table file. For example, enter `bolt_symbol_table.txt`.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_USER_DEFINED_PARAMETRIC_PROFILE_SEPARATORS

Category

Profiles

Use this advanced option to define additional separators to separate dimensions in the names of parametric profiles. The separators can consist of multiple characters.

Separate the values with commas, for example `GA,ABC`.

WARNING Follow these rules in naming:

- Use uppercase letters in separator names.
- Do not enter digits, commas, or special characters in separator names.
- Do not start separator names with a dash (-) or full stop (.).
- Do not start separator names with an inch separator (", ', /) when using imperial units.

In addition to these characters, Tekla Structures always recognizes the standard separator characters X, *, -, and /, and also the character defined by the advanced option XS_PARAMETRIC_PROFILE_SEPARATOR (page 338).

XS_USER_SETTINGS_DIRECTORY

Category

This advanced option must be set in an initialization (.ini) file. It is system-specific.

This advanced option is set as a Windows environment variable in the Windows system properties.

The path to the folder that contains the `user.ini` file and the `options.bin` file.

The default value is `%XSUSERDATADIR%\UserSettings\`

See also

XSUSERDATADIR (page 489)

1.21 Advanced options - V

XS_VALID_CHARS_FOR_ASSEMBLY_FAMILY_POSITION_NUMBERS

Category

Numbering

Specifies valid letters for assembly family position numbers. You must specify all the valid letters in this advanced option. By default, the letters A- Z are valid.

For example, you might not want to use D, because it is easy to mix up with O and 0. In this case you would enter letters A - Z but leave out D.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_ASSEMBLY_FAMILY_POSITION_NUMBER_FORMAT_STRING (page 83)

XS_VALID_CHARS_FOR_ASSEMBLY_FAMILY_QUALIFIER

Category

Numbering

Specifies valid letters for the assembly family number qualifier. You must specify all the valid letters in this advanced option. By default, the letters A - Z are valid.

For example, you might not want to use D, because it is easy to mix it up with O and 0. In this case you would enter letters A - Z but leave out D.

This advanced option is model specific and the setting is saved in the options database.

Example

```
XS_VALID_CHARS_FOR_ASSEMBLY_FAMILY_QUALIFIER=GHJKL
```

See also

XS_ASSEMBLY_FAMILY_POSITION_NUMBER_FORMAT_STRING (page 83)

XS_VALID_CHARS_FOR_ASSEMBLY_MULTI_NUMBERS

Category: Numbering

Use to specify the valid letters for assembly multinumbers. You must specify all the valid letters in this advanced option. By default, letters A - Z are valid.

This advanced option is model specific and the setting is saved in the options database.

Example

```
XS_VALID_CHARS_FOR_ASSEMBLY_MULTI_NUMBERS=ABEG
```

See also

XS_USE_MULTI_NUMBERING_FOR (page 472)

XS_ASSEMBLY_MULTI_NUMBER_FORMAT_STRING (page 85)

XS_MIN_NUMBER_OF_ASSEMBLY_MULTI_CHARACTERS (page 318)

XS_VALID_CHARS_FOR_PART_MULTI_NUMBERS (page 492)

XS_VALID_CHARS_FOR_ASSEMBLY_POSITION_NUMBERS

Category

Numbering

Specifies valid characters for assembly position numbers. Enter all the valid letters, for example, ABEG. By default, letters A - Z are valid.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_ASSEMBLY_POSITION_NUMBER_FORMAT_STRING (page 87)

XS_VALID_CHARS_FOR_PART_MULTI_NUMBERS

Category: Numbering

Use to specify the valid letters for part multinumbers. You must specify all the valid letters in this advanced option. By default, letters a - z are valid.

This advanced option is model specific and the setting is saved in the options database.

Example

```
XS_VALID_CHARS_FOR_PART_MULTI_NUMBERS=abeg
```

See also

XS_USE_MULTI_NUMBERING_FOR (page 472)

XS_PART_MULTI_NUMBER_FORMAT_STRING (page 340)

XS_MIN_NUMBER_OF_PART_MULTI_CHARACTERS (page 319)

XS_VALID_CHARS_FOR_ASSEMBLY_MULTI_NUMBERS (page 491)

XS_VALID_CHARS_FOR_PART_POSITION_NUMBERS

Category

Numbering

Use to specify the valid characters for part position numbers. Enter all the valid letters. For example, ABEG. By default letters A - Z are valid.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_PART_POSITION_NUMBER_FORMAT_STRING (page 340)

XS_VALID_CHARS_FOR_REBAR_SUB_ID_WITH_LETTERS

Category: Numbering

Use this advanced option to specify the valid letters for the reinforcing bar identifiers when they are shown by SUB_ID_WITH_LETTERS (page 598). Enter all the valid letters, for example, ABEG. By default, letters A–Z are valid.

This advanced option is model specific and the setting is saved in the options database.

See also

XS_REBARSET_TAPERED_REBAR_POSITION_NUMBER_FORMAT_STRING (page 379)

XS_VIEW_DIM_LINE_COLOR

Category

Model View

Use to change the dimension line color in model views. Define the color using RGB values:

`<value for red> <value for green> <value for blue>`.

Separate the values with spaces. Define the values on a scale of 0 to 1. The default values are `1.0 0.0 1.0`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

Example

RGB value	Color
`1.0 1.0 1.0`	White
`1.0 0.0 0.0`	Red
`0.0 1.0 0.0`	Green
`0.0 0.0 1.0`	Blue
`1.0 1.0 0.0`	Yellow

XS_VIEW_DIM_TEXT_COLOR

Category

Model View

Use to change the dimension text color in model views. Define the color using RGB values:

`<value for red> <value for green> <value for blue>`.

Separate the values with spaces. Define the values on a scale of 0 to 1. The default values are `0.0 0.0 0.0`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

Example

RGB value	Color
`1.0 1.0 1.0`	White
`1.0 0.0 0.0`	Red
`0.0 1.0 0.0`	Green
`0.0 0.0 1.0`	Blue
`1.0 1.0 0.0`	Yellow

XS_VIEW_FAST_BOLT_COLOR

Category

Model View

Use to define the color of bolts in model views when you are using the representation option **Fast**. Define the color using RGB (Red Green Blue) values. The scale is from `0` to `1`. Separate the numbers with spaces. The default color is white `1.0 1.0 1.0`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

Example

To change the color to black, enter `0.0 0.0 0.0`.

XS_VIEW_FRAMES_VISIBLE

Category: Drawing View

Set this advanced option to `FALSE` to hide drawing view frames until you move the mouse pointer over the view. This is the default value.

When you set this advanced option to `TRUE`, drawing view frames are visible all the time.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_VIEW_FREE_MEASURE_PLANE

Category

Model View

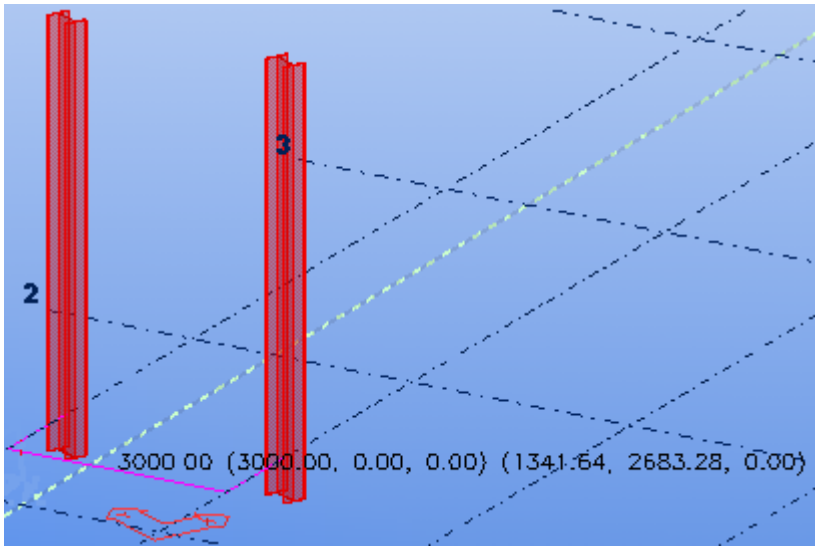
Use to define the plane where the results of free measure are shown. You can have the distances displayed in the local and/or global coordinate system.

The possible values are `VIEW`, `WORK` and `BOTH`. The default value is `VIEW`.

NOTE If you set the advanced option to `BOTH`, only one value is shown if the values are identical.

Example

In the following example, the advanced option has been set to `BOTH`:



XS_VIEW_HEIGHT

Set this advanced option in the `user.ini` file located in `..\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`.

Use to define the default height of model views. Enter the value in pixels.

See also

XS_VIEW_WIDTH (page 498)

XS_VIEW_PART_LABEL_COLOR

Category

Model View

Use to change the part label color in model views. Define the color using RGB values:

`<value for red> <value for green> <value for blue>`.

Separate the values with spaces. Define the values on a scale of 0 to 1. The default is black `0.0 0.0 0.0`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`.

Example

RGB value	Color
`1.0 1.0 1.0`	White
`1.0 0.0 0.0`	Red
`0.0 1.0 0.0`	Green
`0.0 0.0 1.0`	Blue
`1.0 1.0 0.0`	Yellow

XS_VIEW_POSITION_X

Category

Model View

Use to define the default horizontal position of view windows. Origin is in the top left corner of the Tekla Structures or client window. Enter the position in pixels.The default value is `10`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_VIEW_POSITION_Y

Category

Model view

Use to define the default vertical position of view windows. Origin is in the top left corner of the Tekla Structures or client window. Enter the position in pixels. The default value is `10`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_VIEW_TITLE_FONT

Category

Drawing View

Use to specify the font for view direction marks. The default value is Arial. If this advanced option is not set, Tekla Structures uses the font specified for `XS_DEFAULT_FONT`.

TIP If you want to change the view label font, go to **View Properties** --> **Mark Contents** and change the font.

See also

XS_DEFAULT_FONT (page 144)

XS_VIEW_WIDTH

Set this advanced option in the `user.ini` file located in `..\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`.

Use to define the default width of model views. Enter the value in pixels.

See also

XS_VIEW_HEIGHT (page 496)

XS_VISUALIZE_VIEW_IN_ANOTHER_VIEWS

Category

Drawing View

Set this advanced option to `TRUE` if you want to highlight the view boundary of the selected view in another view. If you do not want to highlight the view boundary in another view, set this advanced option to `FALSE`. The default value is `TRUE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_VISUALIZE_VIEW_IN_FATHER_VIEW_ONLY

Category

Drawing View

Set this advanced option to `TRUE` if you want to visualize the section view and detail view boundary boxes only in the view where the section mark or the detail mark is located. If you set this advanced option to `FALSE`, the view boundary boxes are visualized in all of the views where this is possible and the boundary box fits inside the view to some extent. The default value is `TRUE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

XS_VISUALIZE_VIEW_IN_ANOTHER_VIEWS (page 498)

XS_VISUALIZE_VIEW_NEIGHBOUR_PART_EXTENSION

Category

Drawing View

Set this advanced option to `TRUE` to show neighbor part extensions in drawing views. If you set this advanced option to `FALSE`, the neighbor part extensions are not shown. The default value is `TRUE`.

When you select a view, the view extension for neighbor parts for that view is also shown in other views.

NOTE If neighbor parts are hidden by setting **Neighbor parts** to **None** in the **Neighbor Part Properties** dialox box, neighbor part extensions are not displayed even if you set this advanced option to `TRUE`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

See also

1.22 Advanced options - W

XS_WARP_MAX_ANGLE_BETWEEN_CS

Category

Concrete Detailing

Use to define the maximum angle between adjacent polygonal parts of the warped shape.

Enter the value in degrees. You get best results if you use values between 0.5 - 10.0. The default value eis 0.5.

This advanced option is model specific and the setting is saved in the options database.

XS_WARP_MAX_DEVIATION

Category

Concrete Detailing

Use to define the maximum difference between real warped shape and the polygonal warped shape in the model.

Enter the value in millimeters. You get best results if you use values between 5.0 -100.0. The default value is 10.0.

This advanced option is model specific and the setting is saved in the options database.

XS_WELD_FILTER_TYPE

Category

Welds

Use to define how Tekla Structures filters weld types.

- `EXACT`: Tekla Structures filters the welds equal to the default size in the **Welding properties** dialog box
- `MIN`: Tekla Structures filters all welds equal to or smaller than the default size in the **Welding properties** dialog box. This is the default value.

This advanced option is model specific and the setting is saved in the options database.

See also

Model weld mark properties in drawings (page 753)

XS_WELD_FONT

Category

Drawing Properties

Use to specify the font for weld text. The default value is Arial. If this advanced option is not set, Tekla Structures uses the default font defined for `XS_DEFAULT_FONT`.

See also

XS_DEFAULT_FONT (page 144)

XS_WELDING_LENGTH_TOLERANCE

Category

Welds

Use to specify the minimum edge length Tekla Structures should take into account when searching for a location for a weld. The default value is 30 mm.

This advanced option is model specific and the setting is saved in the options database.

XS_WELDING_TOUCH_TOLERANCE

Category: **Welds**

Use to define the maximum possible gap between two parts welded together when copying secondary parts. The weld is copied together with the part if the copied part is within the defined tolerance from the main part. Otherwise, the part is copied without the weld.

The default value is 30 mm.

This advanced option is model specific and the setting is saved in the options database.

XS_WELD_LENGTH_CC_SEPARATOR_CHAR

Category: Welds

Use this advanced option to set the separator character used in the weld mark between the weld length and the pitch (center-to-center spacing) of weld

segments. Enter @ to define the separator character according to the AISC standard (3@12). Enter - to define the separator character according to the ISO standard (100-300). The default value is -.

This advanced option is model specific and the setting is saved in the options database.

See also

Drawing weld mark properties (page 750)

XS_WELD_NUMBER_FORMAT

Category

Marking: General

Use to define the weld number format.

This advanced option is model specific and the setting is saved in the options database.

Example

In `XS_WELD_NUMBER_FORMAT=W%3.3d`:

- `W` is the prefix. The rest of the string defines the number format.
- The first number defines the minimum field width.
- The second number defines the minimum quantity of numbers to display.
- % and d (integer value) indicate the format.

See also

XS_JOINT_NUMBER_FORMAT (page 291)

XS_WORKING_POINTS_VALID_ALSO_OUTSIDE_PART

Category

Dimensioning: Parts

Set this advanced option to `TRUE` to also draw reference dimensions for points outside part end points. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

XS_ZERO_POINT_SYMBOL_OLD_WAY

Category

Dimensioning: General

Set this advanced option to `TRUE` to use RD marks containing a circle rather than the text `RD` to indicate the zero point of dimensions when you use US Absolute dimension types. By default, RD marks contain the text `RD`. The default value is `FALSE`.

This advanced option is model specific and the setting is saved in the options database.

1.23 Advanced options - Z

XS_ZOOM_STEP_RATIO

Category

Model View

Use to configure the **Zoom in** and **Zoom out** commands. The default value is `0.25`. Increase this value to zoom more with a single mouse click.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_ZOOM_STEP_RATIO_IN_MOUSEWHEEL_MODE

Category

Model View

Set the zoom ratio when you are scrolling using the middle mouse button. Enter a decimal value. Increase the value to zoom more with a single mouse click. The default value is `0.05`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

XS_ZOOM_STEP_RATIO_IN_SCROLL_MODE

Category

Model View

Set the zoom ratio when scrolling and holding down the wheel. Enter a decimal value. Increase the value to zoom more with a single mouse click. The default value is `0.01`.

This advanced option is user specific and the setting is saved in `options.bin` under the user folder, for example, in `C:\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UserSettings`. Restart Tekla Structures to activate the new value.

2 Template attributes in drawing and report templates

You can use template attributes in drawing and report templates. When you open a drawing or create a report, Tekla Structures uses the attributes and formulas to calculate and display information from the model database. This could, for example, include assembly weight or cover area.

The template attributes available in a template row definition depend on the content type of the row. Content types are object types in the product database.

Descriptions of template attributes are listed in alphabetical order, click the letter in the table of contents to show all the attributes beginning with that letter.

2.1 Template attributes - A

ACN

Shows control numbers.

For more information on control numbers, see and .

ACTIVE_DESIGN_CODE

Shows the active design code of material.

ADDED_TO_POUR_UNIT

Shows whether an object is added to a pour unit, and how it was added.

Use with the following content types:

- ASSEMBLY
- BOLT
- CAST_UNIT (only precast, not cast-in-place cast units)
- MESH
- REBAR
- REBAR_ASSEMBLY
- SINGLE_REBAR
- SINGLE_STRAND
- STRAND
- STUD

The possible values are:

- 0: The object is not added to any pour unit, or has been modified after the pour units were calculated the last time.
- 1: The object was manually added to the pour unit using the **Add to pour unit** command.
- 2: The object was automatically added to the pour unit using the **Calculate pour units** command.

ADDRESS

Shows the address entered in the **Project properties** in **File menu** --> **Project properties**.

ALIAS_NAME1 ... 3

Alias name of the material.

Use for part and main part material attributes in ASSEMBLY and PART content types.

ANALYSIS_MODEL_NAME

Shows the name of the analysis model in which the rigid link is included.

Use with the `ANALYSIS_RIGID_LINK` content type.

ANG_S, ANG_T, ANG_U, ANG_V

Show bending angles of reinforcing bars based on the mappings in the `rebar_schedule_config.inp` file, located in the `..\ProgramData\Trimble\Tekla Structures\<version>\environments\<environment>\system` folder. These mappings are environment-specific by default. You can modify them to suit your company or project needs.

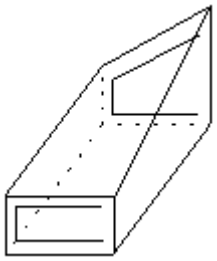
See also

ANG_U_MAX, ANG_U_MIN, ANG_V_MAX, ANG_V_MIN (page 507)

DIM_A ... DIM_G, DIM_H1, DIM_H2, DIM_I, DIM_J, DIM_K1, DIM_K2, DIM_L, DIM_O, DIM_R, DIM_R_ALL, DIM_TD, DIM_WEIGHT, DIM_X, DIM_Y (page 545)

ANG_U_MAX, ANG_U_MIN, ANG_V_MAX, ANG_V_MIN

Shows the minimum and maximum bending angles of reinforcing bars or meshes in tapered cross sections. See the example below:



APPROVED_BY

The **Approved by** information of the revision from the **Revision Handling** dialog box.

AREA

Shows the following information:

- For plate type catalog profiles, any parametric profiles and any catalog profiles with **Cover area** property not defined, shows the total net area of all surfaces.
- For other types of catalog profiles with **Cover area** property defined, shows the gross total surface area.

  The area is calculated using the extreme length and profile cover area per meter (value defined in the profile catalog). The cross area on profile ends, cuts and fittings are not taken into account.

See also

AREA_GROSS (page 509)

AREA_NET (page 510)

AREA_FORM_TOP, AREA_FORM_BOTTOM, AREA_FORM_SIDE

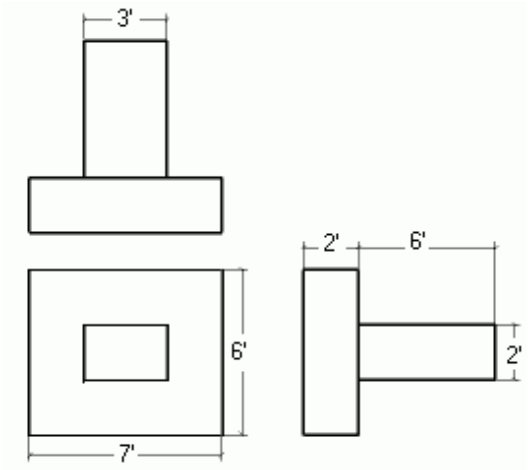
These template attributes show the area of faces whose normal vector points in the following directions:

- Top of form (`AREA_FORM_TOP`)
- Bottom of form (`AREA_FORM_BOTTOM`)
- Form sides (`AREA_FORM_SIDE`)

Use these template attributes with the `CAST_UNIT` content type to report the formwork areas of precast cast units.

For assemblies and cast units, the main part local up direction dictates the form up/bottom/sides directions. Faces which are inclined less than 5 degrees are counted in the top and bottom areas. Faces which are skew => 85 degrees are counted in the side areas. Faces which are exactly 45 degrees against main global or local axes, are not counted to any direction.

Steel embeds are ignored when calculating the `AREA_FORM_...` values of cast units.



AREA_FORM_TOP = 42 sq.ft.

AREA_FORM_BOTTOM = 42 sq.ft.

AREA_FORM_SIDE = 2*(2*7 + 3*6) + 2*(2*6 + 2*6) = 64 + 48 = 112 sq.ft.

To report the formwork areas of cast-in-place cast units, use the template attributes (page 509) AREA_FORM_TOP_GLOBAL, AREA_FORM_BOTTOM_GLOBAL, and AREA_FORM_SIDE_GLOBAL with the CAST_UNIT content type. With these . . . _GLOBAL attributes, **Top-in-form face** settings have no effect.

AREA_FORM_TOP_GLOBAL, AREA_FORM_BOTTOM_GLOBAL, AREA_FORM_SIDE_GLOBAL

These template attributes show the area of faces whose normal vector points in the following directions in the global coordinate system:

- Top of form (AREA_FORM_TOP_GLOBAL)
- Bottom of form (AREA_FORM_BOTTOM_GLOBAL)
- Form sides (AREA_FORM_SIDE_GLOBAL)

Use these template attributes with the CAST_UNIT content type to report the formwork areas of cast-in-place cast units. These attributes and areas are not dependent on the **Top-in-form face** settings.

To report the formwork areas of precast cast units, use the template attributes (page 508) AREA_FORM_TOP, AREA_FORM_BOTTOM, and AREA_FORM_SIDE.

AREA_GROSS

For profiles this field shows the same result as AREA (page 507). For plates it shows the square area (extreme length multiplied by extreme width) needed to include the entire plate. For other objects it shows a zero.

AREA_NET

For parts this field shows the net surface area that forms the actual area of the fabricated part. For other objects it shows a zero.

AREA_PER_TONS

Shows AREA/WEIGHT x 1000.

AREA_PGX, AREA_NGX, AREA_PGY, AREA_NGY, AREA_PGZ, AREA_NGZ

Shows the area of faces whose normal vector points to the positive or negative direction of the following global axes:

Attribute	Direction
AREA_PGX	Positive direction of global X-axis
AREA_NGX	Negative direction of global X-axis
AREA_PGY	Positive direction of global Y-axis
AREA_NGY	Negative direction of global Y-axis
AREA_PGZ	Positive direction of global Z-axis
AREA_NGZ	Negative direction of global Z-axis

Also faces whose normal vector is located in less than 45 degree angle to global axis are also included in the area. Faces exactly in 45 degree angle are not included in any global direction.

AREA_PLAN

For parts this field shows the total upper surface area (perpendicular to the global Z-axis).

ASSEMBLY content type

- Shows the total upper surface area (perpendicular to the global Z-axis) of the parts included in an assembly.

AREA_PROJECTION_GXY_GROSS, AREA_PROJECTION_GXZ_GROSS, AREA_PROJECTION_GYZ_GROSS

Shows the area of the "shadow" of a part, assembly, or cast unit at the following global planes:

- XY-plane
- XZ-plane
- YZ-plane

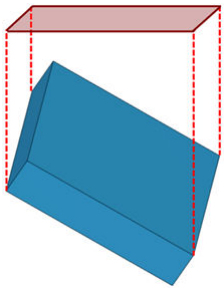
Restrictions

- Areas are calculated always in net areas (holes are taken into account) even when gross is requested.
- Overlapping faces are counted twice.

AREA_PROJECTION_GXY_NET, AREA_PROJECTION_GXZ_NET, AREA_PROJECTION_GYZ_NET

Shows the net area of the "shadow" of a part, assembly, or cast unit at the following global planes:

- XY-plane
- XZ-plane
- YZ-plane



AREA_PROJECTION_XY_GROSS, AREA_PROJECTION_XZ_GROSS, AREA_PROJECTION_YZ_GROSS

Shows the area of the "shadow" of a part, assembly, or cast unit at its local planes:

- XY-plane
- XZ-plane

- YZ-plane

AREA_PROJECTION_XY_NET, AREA_PROJECTION_XZ_NET, AREA_PROJECTION_YZ_NET

Shows the net area of the "shadow" of a part, assembly, or cast unit at its local planes:

- XY-plane
- XZ-plane
- YZ-plane

AREA_PX, AREA_NX, AREA_PY, AREA_NY, AREA_PZ, AREA_NZ

Shows the area of faces whose normal vector points to the positive or negative direction of the following local axes:

Attribute	Direction
AREA_PX	Positive direction of local X-axis
AREA_NX	Negative direction of local X-axis
AREA_PY	Positive direction of local Y-axis
AREA_NY	Negative direction of local Y-axis
AREA_PZ	Positive direction of local Z-axis
AREA_NZ	Negative direction of local Z-axis

ASSEMBLY.LOCK_PERMISSION

Shows the effective permission for the assembly. Options are **ALL** or **NONE**.

See also

ASSEMBLY.OBJECT_LOCKED (page 512)

ASSEMBLY.OWNER_ORGANIZATION (page 513)

ASSEMBLY.OBJECT_LOCKED

Shows the value of the object lock. The value options are **Yes**, **No**, and **Organization**.

The object lock status can be modified in the **Object locks** dialog box.

See also

ASSEMBLY.OWNER_ORGANIZATION (page 513)

ASSEMBLY.LOCK_PERMISSION (page 512)

ASSEMBLY.OWNER_ORGANIZATION

Shows the name of the organization that owns the assembly lock. The organization is based on the Windows account.

See also

ASSEMBLY.OBJECT_LOCKED (page 512)

ASSEMBLY.LOCK_PERMISSION (page 512)

ASSEMBLY_BOTTOM_LEVEL

Shows the bottom level of an assembly. Bottom level takes the unit and accuracy from the `MarkDimensionFormat.dim` file. You can modify the settings in `MarkDimensionFormat.dim` in the **Dimension Properties** dialog box in a drawing.

You can use this attribute as a user-defined attribute also in part marks and associative notes.

NOTE This attribute returns the value as text, so you cannot use formulae with this attribute. Use ASSEMBLY_BOTTOM_LEVEL_UNFORMATTED (page 514) instead.

See also

XS_DRAWING_IGNORE_ZERO_LEVELS_IN_PART_MARKS (page 208)

ASSEMBLY_BOTTOM_LEVEL_GLOBAL

Shows the bottom level of an assembly by global axis. The bottom level takes the unit and accuracy from the `MarkDimensionFormat.dim` file. You can

modify the settings in `MarkDimensionFormat.dim` in the **Dimension Properties** dialog box in a drawing.

You can use this attribute as a user-defined attribute in part marks and associative notes, and also in reports and templates.

See also

ASSEMBLY_BOTTOM_LEVEL_GLOBAL_UNFORMATTED

Shows the bottom level of an assembly by global axis. Unformatted level returns the bottom levels as a length in `mm` so you can format them and include them into formulas in templates.

You can use this attribute as a user-defined attribute also in part marks and associative notes.

ASSEMBLY_BOTTOM_LEVEL_UNFORMATTED

Shows the unformatted bottom level of an assembly. Unformatted level returns the bottom levels as a length in `mm` so you can format them and include them into formulas in templates.

You can use this attribute as a user-defined attribute also in part marks and associative notes.

NOTE Unlike the `ASSEMBLY_BOTTOM_LEVEL` attribute, the `ASSEMBLY_BOTTOM_LEVEL_UNFORMATTED` attribute cannot be formatted through the `MarkDimensionFormat.dim` file.

See also

ASSEMBLY_DEFAULT_PREFIX

Shows the default value for the assembly prefix defined in the part properties.

ASSEMBLY_PLWEIGHT

Shows the weight of plates attached to an assembly. For other objects it shows a zero.

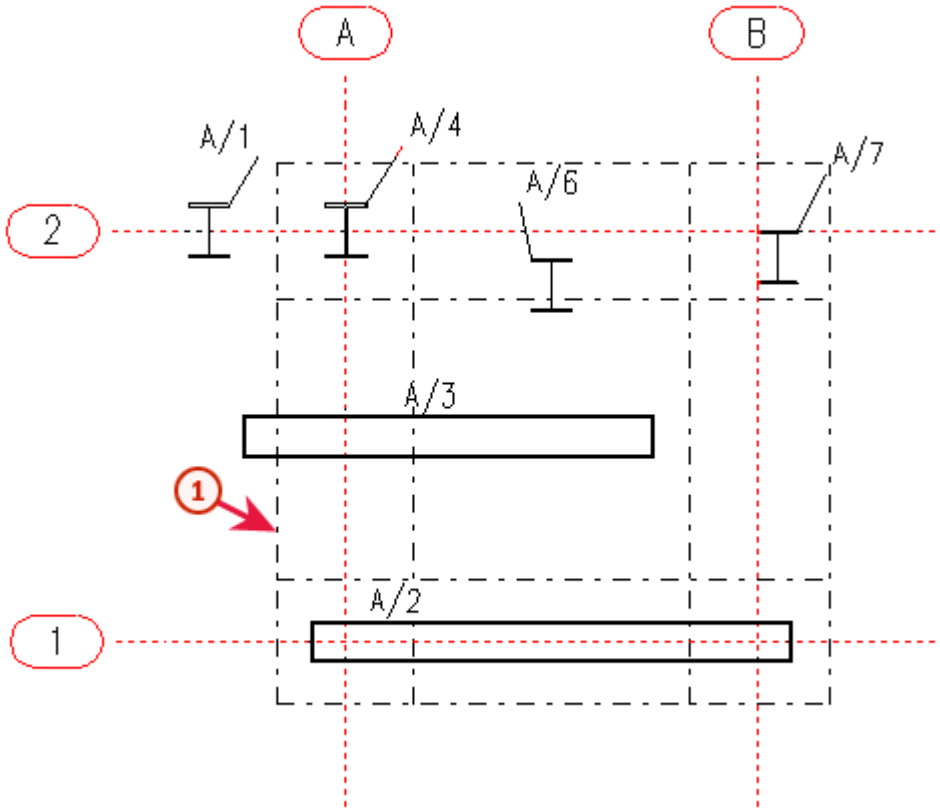
ASSEMBLY_POS

Shows the assembly position number. For parts `ASSEMBLY_POS` shows the assembly position number of the assembly that contains the part. For reinforcement objects `ASSEMBLY_POS` shows the assembly position number of the rebar assembly that contains the reinforcement object. For bolts the field is blank.

ASSEMBLY_POSITION_CODE

Shows the assembly position code. The code identifies the grid position. The position of the objects is calculated based on the closest grid.

Assembly	Code
A/1	<A/2
A/2	A-B/1
A/3	<A-B/1-2
A/4	A/2
A/6	A-B/1-2
A/7	B/2



1 TOLERANCE LINE

The position code consists of grid line labels in the x and y directions (alternatively in the z direction). If an assembly begins or ends outside the first or last grid line, a < or > character is included in the position code. For example, if an assembly begins outside the A grid line, this field shows:

<A/2

If an assembly is completely within a tolerance distance (by default 500 mm) of grid line A, the position code is the label of that grid line: A.

If the assembly is partially or entirely outside the tolerance distance, the code is a combination of grid labels: A-B.

To change the default tolerance distance, set the advanced option `XS_ASSEMBLY_POSITION_CODE_TOLERANCE=750` (for example).

To include the Z orientation in the code, set the advanced option `XS_ASSEMBLY_POSITION_CODE_3D` to TRUE. The code would be similar to: <A-B/1-2/1-+1000

Tekla Structures selects the grid to use as follows:

1. Tekla Structures checks the location of the assembly.

2. If it is located inside several grids, Tekla Structures checks whether the assembly is parallel to grid lines or the plane.
3. If there are several parallel grids, Tekla Structures selects the closest.

ASSEMBLY_PREFIX

Shows the assembly prefix, defined in the part or assembly properties.

ASSEMBLY_SERIAL_NUMBER

Shows the assembly number without prefix and separator.

ASSEMBLY_START_NUMBER

Shows the assembly start number.

See also

ASSEMBLY_TOP_LEVEL

Shows the top level of an assembly. Top level takes the unit and accuracy from the `MarkDimensionFormat.dim` file. You can modify the settings in `MarkDimensionFormat.dim` in the **Dimension Properties** dialog box in a drawing.

You can use this attribute as a user-defined attribute also in part marks and associative notes.

NOTE This attribute returns the value as text, so you cannot use formulae with this attribute. Use ASSEMBLY_TOP_LEVEL_UNFORMATTED (page 518) instead.

See also

XS_DRAWING_IGNORE_ZERO_LEVELS_IN_PART_MARKS (page 208)

ASSEMBLY_TOP_LEVEL_GLOBAL

Shows the top level of an assembly by global axis. The top level takes the unit and accuracy from the `MarkDimensionFormat.dim` file. You can modify the

settings in `MarkDimensionFormat.dim` in the **Dimension Properties** dialog box in a drawing.

You can use this attribute as a user-defined attribute in part marks and associative notes, and also in reports and templates.

See also

ASSEMBLY_TOP_LEVEL_GLOBAL_UNFORMATTED

Shows the top level of an assembly by global axis. Unformatted level returns the top levels as a length in `mm` so you can format them and include them into formulas in templates.

You can use this attribute as a user-defined attribute also in part marks and associative notes.

ASSEMBLY_TOP_LEVEL_UNFORMATTED

Shows the unformatted top level of an assembly. Unformatted level returns the top levels as a length in `mm` so you can format them and include them into formulas in templates.

You can use this attribute as a user-defined attribute also in part marks and associative notes.

NOTE Unlike the `ASSEMBLY_TOP_LEVEL` attribute, the `ASSEMBLY_TOP_LEVEL_UNFORMATTED` attribute cannot be formatted through the `MarkDimensionFormat.dim` file.

ATTACHED_TO

Shows whether a surface is attached to a part or to a pour.

The attribute returns 0 if the surface is attached to a part, and 1 if the surface is attached to a pour.

axial1, axial2

Shows the values entered in the **Tension, T** box on the **End codes** tab in the user-defined attributes dialog box of the part. `axial1` shows the value in the **Start** box and `axial2` in the **End** box.

2.2 Template attributes - B

BOLT_COUNTERSUNK

Use to check or show if a bolt is countersunk. The attribute returns the value `1` for countersunk bolts, otherwise it returns `0`.

See also

HEAD_TYPE (page 554)

BOLT_EDGE_DISTANCE

Shows the edge distance of a bolt.

BOLT_EDGE_DISTANCE_MIN

Shows the edge distance multiplied by the coefficient set in the modeling settings in **File menu** --> **Settings** --> **Options** --> **Components** .

BOLT_FULL_NAME

Shows the name of a bolt defined in the bolt catalog, without the standard.

For objects other than bolts, the field shows a blank.

See also

BOLT_SHORT_NAME (page 520)

BOLT_MATERIAL_LENGTH

For bolts this field shows the total thickness of the connected material.

BOLT_NPARTS

For bolts this field shows the number of connected parts.

BOLT_SHORT_NAME

Shows the name of the washer, bolt, nut or screw in a short format.

See also

BOLT_FULL_NAME (page 519)

BOLT_STANDARD

As for TYPE (page 602).

BOLT_THREAD_LENGTH

Shows the length of the threaded part of the bolt shaft.

BOTTOM_LEVEL

Shows the bottom level of a single part, cast unit, assembly, part of a connection or a pour object.

Bottom level takes the unit and accuracy from the `MarkDimensionFormat.dim` file. You can modify the settings in `MarkDimensionFormat.dim` in the **Dimension Properties** dialog box in a drawing.

You can use this attribute as a user-defined attribute also in part marks and associative notes.

NOTE This attribute returns the value as text, so you cannot use formulae with this attribute. Use BOTTOM_LEVEL_UNFORMATTED (page 521) instead.

BOTTOM_LEVEL_GLOBAL

Shows the bottom level of a single part, cast unit, assembly, part of a connection or a pour object by global axis. `BOTTOM_LEVEL_GLOBAL` takes the unit and accuracy from `MarkDimensionFormat.dim`. You can modify the

settings in MarkDimensionFormat.dim in the **Dimension Properties** dialog box in a drawing.

You can use this attribute as a user-defined attribute in part marks and associative notes, and also in reports and templates.

BOTTOM_LEVEL_GLOBAL_UNFORMATTED

Shows the bottom level of a single part, cast unit, assembly, part of a connection or a pour object. BOTTOM_LEVEL_GLOBAL_UNFORMATTED returns the bottom levels as a length in mm so you can format them and include them into formulas in templates. This attribute gives level information by the global axis.

You can use this attribute as a user-defined attribute also in part marks and associative notes.

BOTTOM_LEVEL_UNFORMATTED

Shows the unformatted bottom level of a single part, cast unit, assembly, part of a connection or a pour object. BOTTOM_LEVEL_UNFORMATTED returns the bottom levels as a length in mm so you can format them and include them into formulas in templates.

You can use this attribute as a user-defined attribute also in part marks and associative notes.

NOTE Unlike the BOTTOM_LEVEL attribute, the BOTTOM_LEVEL_UNFORMATTED attribute cannot be formatted through the MarkDimensionFormat.dim file.

BOUNDING_BOX_xxx

The following template attributes give the bounding box of the objects as X, Y or Z minimum or maximum distances from the absolute zero (0,0,0):

- BOUNDING_BOX_MIN_X
- BOUNDING_BOX_MAX_X
- BOUNDING_BOX_MIN_Y
- BOUNDING_BOX_MAX_Y

- BOUNDING_BOX_MIN_Z
- BOUNDING_BOX_MAX_Z

These attributes are available for parts, assemblies, cast units, reference models and reference objects.

BUILDER

Shows the builder's name defined in the **Project properties** in **File menu** --> **Project properties**.

2.3 Template attributes - C

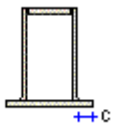
cambering

Shows the value entered in the **Camber** box on the **Parameters** tab in the user-defined attributes dialog box of the part.

See also

CANTILEVER

Shows the length of a protruding part of a profile. Below is an example of a welded box profile:



See also

PROFILE (page 582)

CAST_UNIT_BOTTOM_LEVEL

Shows the bottom level of a cast unit.

You can use this attribute as a user-defined attribute also in part marks and associative notes.

See also

XS_DRAWING_IGNORE_ZERO_LEVELS_IN_PART_MARKS (page 208)

CAST_UNIT_HEIGHT_ONLY_CONCRETE_PARTS

Shows the height of a cast unit including all concrete parts.

CAST_UNIT_HEIGHT_ONLY_PARTS

Shows the height of a cast unit, including all concrete parts, steel parts and parts made of miscellaneous material.

CAST_UNIT_HEIGHT_TOTAL

Shows the total height of a cast unit, including all concrete parts, steel parts and parts made of miscellaneous material, reinforcing bars, surface treatments and bolts.

CAST_UNIT_LENGTH_ONLY_CONCRETE_PARTS

Shows the length of a cast unit including all concrete parts.

CAST_UNIT_LENGTH_ONLY_PARTS

Shows the total length of a cast unit, including all concrete parts, steel parts and parts made of miscellaneous material.

CAST_UNIT_LENGTH_TOTAL

Shows the total length of a cast unit, including all concrete parts, steel parts and parts made of miscellaneous material, reinforcing bars, surface treatments and bolts.

CAST_UNIT_POS

Shows the position of a cast unit. The position consists of a prefix and a number.

CAST_UNIT_POSITION_CODE

Shows the position code of a cast unit. The code identifies the grid position. For more information, see ASSEMBLY_POSITION_CODE (page 515).

CAST_UNIT_PREFIX

Shows the cast unit prefix, defined in the part properties.

CAST_UNIT_REBAR_WEIGHT

Shows the weight of reinforcing bars in a cast unit.

CAST_UNIT_SERIAL_NUMBER

Shows the cast unit number without prefix and separator.

CAST_UNIT_TOP_LEVEL

Shows the top level of a cast unit.

You can use this attribute as a user-defined attribute also in part marks and associative notes.

See also

XS_DRAWING_IGNORE_ZERO_LEVELS_IN_PART_MARKS (page 208)

CAST_UNIT_TYPE

Returns the type of the cast unit as text (`Precast` or `Cast in place`).

CAST_UNIT_VERTICAL_POSITION_CODE

Outputs the grid level height of a cast unit, for example +7200. The center of gravity point is used to determine the grid level for the cast unit. If the center of gravity is more than 100 mm away from the grid level, then two grid levels will be output separated with dash: the lower and higher grid levels, for example, +3600-+7200.

See also

ASSEMBLY_POSITION_CODE (page 515)

CAST_UNIT_WIDTH_ONLY_CONCRETE_PARTS

Shows the width of a cast unit including all concrete parts.

CAST_UNIT_WIDTH_ONLY_PARTS

Shows the total width of a cast unit, including all concrete parts, steel parts and parts made of miscellaneous material.

CAST_UNIT_WIDTH_TOTAL

Shows the total width of a cast unit, including all concrete parts, steel parts and parts made of miscellaneous material, reinforcing bars, surface treatments and bolts.

CATALOG_NAME

Shows the reinforcement mesh identifier, for example, 8-200-2350/5000 or Custom Mesh. For standard meshes, shows the mesh name used in the mesh catalog file `mesh_database.inp`.

This identifier is also shown in the **Mesh** box in the **Rebar mesh** properties (page 635), in the **Select mesh** dialog box for standard meshes, and as **Catalog name** in the **Custom component browser**.

See also

Reinforcement mesh properties (page 635)

CC

Shows the center-to-center spacing of evenly-distributed reinforcing bars or a mesh.

CC_CROSS

Shows the center-to-center spacing of crossing bars in a reinforcement mesh.

CC_DIAMETER_xxx

The `CC_DIAMETER_` template attributes show the bar diameters of a reinforcement mesh.

Template attribute	Description
`CC_DIAMETER_CROSS`	Shows all diameters of the crossing bars. For example, `30*8 4*10`.
`CC_DIAMETER_LONG`	Shows all diameters of the longitudinal bars. For example, `5*10 25*8 5*10`.
`CC_DIAMETER_MAX_CROSS`	Shows the largest diameter of the crossing bars.
`CC_DIAMETER_MAX_LONG`	Shows the largest diameter of the longitudinal bars.
`CC_DIAMETER_MIN_CROSS`	Shows the smallest diameter of the crossing bars.

Template attribute	Description
CC_DIAMETER_MIN_LONG	Shows the smallest diameter of the longitudinal bars.

CC_EXACT

Shows the center-to-center spacing of a reinforcing bar group or a mesh.

CC_EXACT_CROSS

Shows all center-to-center spacings of crossing bars in a reinforcement mesh.

CC_EXACT_LONG

Shows all center-to-center spacings of longitudinal bars in a reinforcement mesh.

CC_LONG

Shows the center-to-center spacing of longitudinal bars in a reinforcement mesh.

CC_MAX

Shows the largest center-to-center spacing in reinforcing bar groups or meshes with varied spacing.

CC_MAX_CROSS

Shows the largest center-to-center spacing of crossing bars in reinforcement meshes with varied spacing.

CC_MAX_LONG

Shows the largest center-to-center spacing of longitudinal bars in reinforcement meshes with varied spacing.

CC_MIN

Shows the smallest center-to-center spacing in reinforcing bar groups or meshes with varied spacing.

CC_MIN_CROSS

Shows the smallest center-to-center spacing of crossing bars in reinforcement meshes with varied spacing.

CC_MIN_LONG

Shows the smallest center-to-center spacing of longitudinal bars in reinforcement meshes with varied spacing.

CC_TARGET

Shows the target center-to-center spacing value in reinforcing bar groups, rebar sets bar groups, or meshes.

CHANGES

The `CHANGES` attribute tells the changes occurred in a drawing, for example, if the drawing is changed while it is issued, or if part have been modified. This attribute can be used for adding **Document manager** information about changes in drawing reports. Also **Document manager** has a column **Changes** for this information.

Below is an example of the changes column in **Document manager**.

Name	Changes
STANDARD	Parts modified
GA-drawing	
CAST UNIT	
STANDARD	Drawing was updated
STANDARD	Issued drawing changed
STANDARD	

CHECKED_BY

This attribute gets the value entered in the **Checked By** box on the **Status** tab in the user-defined attributes dialog box of the part or in the assembly properties dialog box. Also shows the value that you have entered in the **Checked By** field in the **Revision Handling** dialog box.

CHECKED_DATE

Shows the value entered in the **Date Checked** box on the **Status** tab in the user-defined attributes dialog box of the part or in the assembly properties dialog box.

CLASS

Only use to set rules in the Template Editor. It shows the string `ASSEMBLY` for assemblies, `PART` for parts, and `BOLT` for bolts, holes, nuts etc. For drawings it shows `DRAWING`, and for revisions it shows `REVISION`.

CLASS_ATTR

Shows the class number of parts, reinforcement, and surfaces.

For assemblies and cast units, `MAINPART.CLASS_ATTR` shows the class number of the main part.

For bolts, welds, and connections, `CLASS_ATTR` can be used for showing the class number of bolted, welded, or connected parts. For example, to show the class number of the bolt main part or the first secondary part, use `MAIN_PART.CLASS_ATTR` or `SECONDARY_1.CLASS_ATTR`.

CODE

Shows the abbreviation code of a surface treatment, for example, TS1 for Tile surface 1.

Surface treatment codes and names are defined in the `product_finishes.dat` file.

See also

SURFACING_NAME (page 599)

COG_X, COG_Y, COG_Z

Shows the coordinates of the center of gravity of assemblies, parts, or welds:

- For parts, assemblies, and cast units the attributes COG_X, COG_Y, and COG_Z return values in global coordinate system.
- For welds the attributes COG_X, COG_Y, and COG_Z return values in local coordinate system (current work plane grid).

These attributes cannot be used in headers or footers.

comment

User-defined attribute **Comment**, defined in the object's user-defined attributes dialog box. For more information on the user-defined attributes in templates and reports, see .

CONCRETE_COVER_FROM_PLANE

Shows the distance from the part surface to the reinforcing bar, perpendicular to the bar plane.

This is the first value entered in the **From plane** box in the **Single rebar** or **Rebar group** properties (page 632).

See also

CONCRETE_COVER_ON_PLANE (page 531)

CONCRETE_COVER_START, CONCRETE_COVER_END (page 531)

CONCRETE_COVER_ON_PLANE

Shows the distance from the part surface to the reinforcing bar on the bar plane.

This is the first value entered in the **On plane** box in the **Single rebar** or **Rebar group** properties (page 632).

To show the minimum or maximum value entered in the **On plane** box, use the following template attributes:

- CONCRETE_COVER_ON_PLANE_MIN
- CONCRETE_COVER_ON_PLANE_MAX

See also

CONCRETE_COVER_FROM_PLANE (page 530)

CONCRETE_COVER_START, CONCRETE_COVER_END (page 531)

CONCRETE_COVER_START, CONCRETE_COVER_END

CONCRETE_COVER_START shows the concrete cover thickness at the first end of the reinforcing bar. CONCRETE_COVER_END shows the concrete cover thickness at the second end of the reinforcing bar.

These are the values entered in the **Start** and **End** boxes in the **Single rebar** or **Rebar group** properties (page 632) when the **Cover thickness** option is selected.

See also

CONCRETE_COVER_ON_PLANE (page 531)

CONCRETE_COVER_FROM_PLANE (page 530)

LEG_LENGTH_START, LEG_LENGTH_END (page 566)

CONN_CODE_END1, CONN_CODE_END2

Shows the values entered in the **Connection code** box on the **End conditions** tab in the user-defined attributes dialog box of the part. CONN_CODE_END1 shows the value in the **Start** box and CONN_CODE_END2 in the **End** box.

CONNECTED_ASSEMBLIES

For bolts this field shows a string containing the position numbers of assemblies of connected parts (e.g. A17 A18 A23). In `ASSEMBLY_BOLT` lists Tekla Structures does not show the position number of the current assembly. Only use this field as an inquiry command for single bolts. For objects other than bolts the field is blank.

CONNECTED_PARTS

Shows a string containing the position numbers of connected parts (e.g. P102 -> P17 P18 P23) for bolts. If the list type is `ASSEMBLY_BOLT`, the first position number is a member of the current assembly. Only use as an inquiry command for single bolts. For objects other than bolts the field is blank.

CONNECTION_CODE

Shows the connection code defined in the connection properties dialog box. Only for use in connection lists.

CONNECTION_DSTV

Shows the DSTV code of the connection in connection lists. This field blank if the connection is not a DSTV connection. Only for use in connection lists.

CONNECTION_ERROR

Shows the error flag of a connection in connection lists. Only for use in connection lists.

The values returned are:

- 1=green connection symbol
- 2=yellow connection symbol
- 3=red connection symbol
- 4=connection did not pass design check

CONNECTION_GROUP

Shows the class of the component, available on the **General** tab in the component dialog box. Only for use in connection lists.

CONNECTION_NUMBER

Shows the number of a connection.

CONNECTION_RUNNING_NUMBER

Shows the running number of a connection. All connections are automatically numbered with a running number.

CONTENTTYPE

Shows the content type of the current row.

See also

COUNTRY

Shows the country entered in the **Project properties** in **File** --> **Project properties** .

COVER_AREA

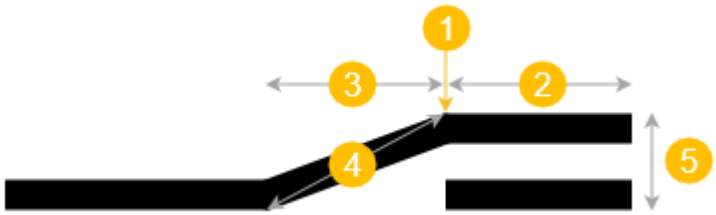
Shows the total cover area of the part profile, or of the main part profile in the assembly or cast unit.

See also

PROFILE (page 582)

CRANK_xxx

Use the following template attributes to show rebar set bar crank information defined by using a rebar set splitter or end detail modifier.



(1) = Location of the splitter

Template attribute	Description
CRANK_SIDE_START CRANK_SIDE_END	Shows to which side of the splitter the crank is created at the start or end of the bar: `Left` or `Right`.
CRANK_ROTATE_START CRANK_ROTATE_END	Shows to which angle the crank is rotated at the start or end of the bar.
CRANK_STRLEN_START CRANK_STRLEN_END	Shows the length of the straight segment of the crank at the start or end of the bar. This is **(2)** in the image above.
CRANK_LENTYPE_START CRANK_LENTYPE_END	Shows the type of the cranked length at the start or end of the bar: `Diagonal ratio`, `Diagonal distance`, `Horizontal ratio`, `Horizontal distance`.
CRANK_RATIO_START CRANK_RATIO_END	Shows the multiplier of the bar diameter that is used to define the cranked length at the start or end of the bar.
CRANK_DIST_START CRANK_DIST_END	Shows the length of the cranked segment at the start or end of the bar. If the cranked length type is `Horizontal distance`, this is **(3)** in the image above. If the cranked length type is `Diagonal distance`, this is **(4)** in the image above.
CRANK_OFFSET_START CRANK_OFFSET_END	Shows the offset distance of the straight segment of the crank at the start or end of the bar. This is **(5)** in the image above.

See also

Splitter properties (page 652)

CREATED_BY

This attribute gets the name of the revision creator.

CROSS_SECTION_AREA

Shows the area (mm2) of a cross section.

See also

PROFILE (page 582)

CURRENT_PHASE

Shows the current phase. Used for filtering parts. You can also use selection filters.

CURVED_SEGMENTS

Returns the number of segments of a curved beam.

CUSTOM.ELEMENT_WEIGHT

This custom template attribute sums up net weights of all cast unit and subassembly parts, but ignores all subassemblies whose main part's `MATERIAL_TYPE` is `STEEL`.

The same weight is wanted to be reported

1. early in the project when only sample elements are detailed but the great majority of the elements is not
2. in the final stage of the project when all elements have been fully detailed

The `CAST_UNIT.WEIGHT` attribute also takes into account the weight of all embedded subassemblies, such as lifting anchors and cable loops. This is not

wanted as the reinforcement and embed weights are already included into a little bit exaggerated concrete density.

For cast units with dense reinforcement, the CUSTOM.ELEMENT_WEIGHT.REINFORCED attribute is more accurate than CUSTOM.ELEMENT_WEIGHT.

CUSTOM.HC_xxx

The following part-specific opening and area calculations are available for Hollowcore slabs. The calculations can be output with custom reports.

The report property names are:

- CUSTOM.HC_GROSS_AREA: This is the gross area calculated by formula L*B , where L is the max length of the slab and B is the width of the original hollow core slab section before any narrow cutting of the slab.
- CUSTOM.HC_INSUL_CUT_L: This is the total linear length of insulation cutting measured along insulation edges where the edge of insulation is not overlapping with exterior edges of the slab.
- CUSTOM.HC_NET_AREA: This is the net area of the hollow core slab. The net area is excluding all openings penetrating.
- CUSTOM.HC_OPENINGS_L: This is the total perimeter length of all openings in the slab. The perimeter is measured along the “shape boundary” of the opening.
- CUSTOM.HC_RECESSES_L: This is the total perimeter of recesses (not fully penetrating the slab thickness). The perimeter is measured along the “shape boundary” of the recess.
- CUSTOM.HC_SAWINGS_END_L: This is the total linear length of skew end sawings in the slab. Please note that the straight ends are not counted to the total sawing length.
- CUSTOM.HC_SAWINGS_END_N: This is the total number of individual sawing lines.
- CUSTOM.HC_SAWINGS_SIDE: This is the total length of sawing parallel to center axis of the slab.

In Template Editor these attributes are located in the CUSTOM subfolder in the **Attribute** dialog box.

CUSTOM.MESH_xxx

The following attributes are available for reinforcement meshes:

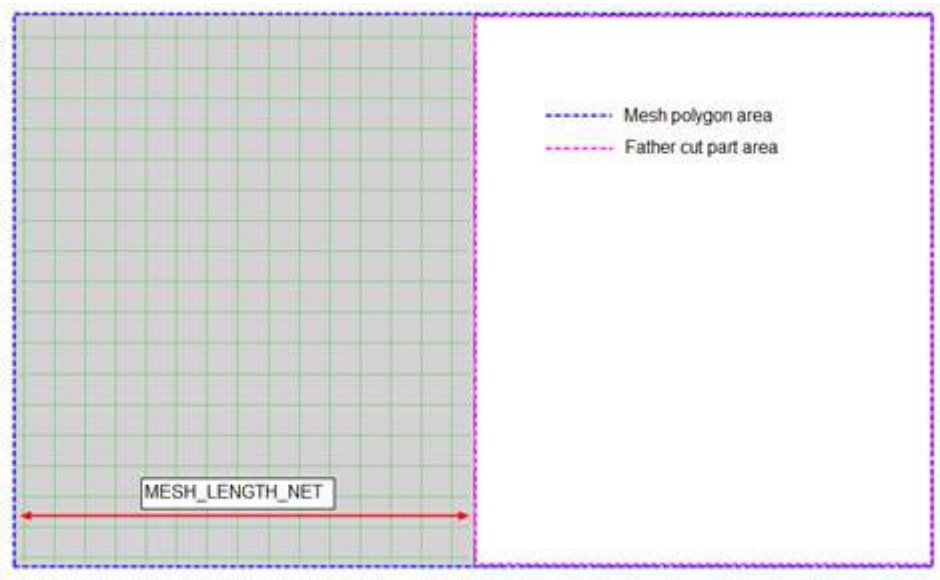
- `CUSTOM.MESH_LENGTH_NET` (distance)
- `CUSTOM.MESH_WIDTH_NET` (distance)
- `CUSTOM.MESH_SIZE_NET` (text)

All these attributes are calculated based on the mesh wires considering all cuts. The net length is always the longer dimension of the mesh and the net width is the shorter. The net size is always expressed based on net length and net width including the text for sizes and spacings.

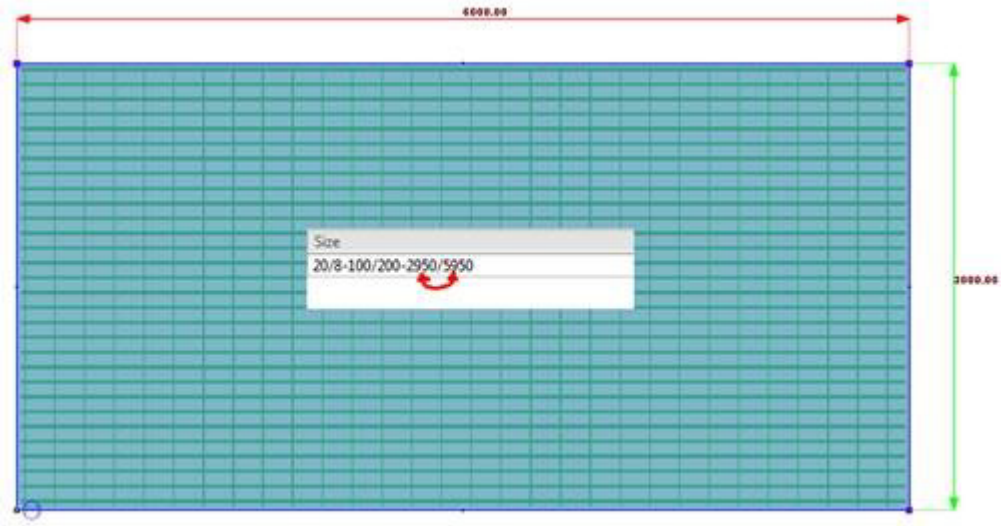
The calculations can be output with custom reports. In Template Editor they are located in the CUSTOM subfolder in the **Attributes** dialog box.

We recommend that you use these attributes instead of any other mesh attributes for size calculations.

Tekla Structures length inquiry gives the whole length, whereas the `MESH_LENGTH_NET` gives the length of the mesh itself.



Tekla Structures size inquiry gives the size so that it gives the height first and the width last, whereas `MESH_SIZE_NET` reports the width first and the height last: 20/8-100/200-**5950/2950** .



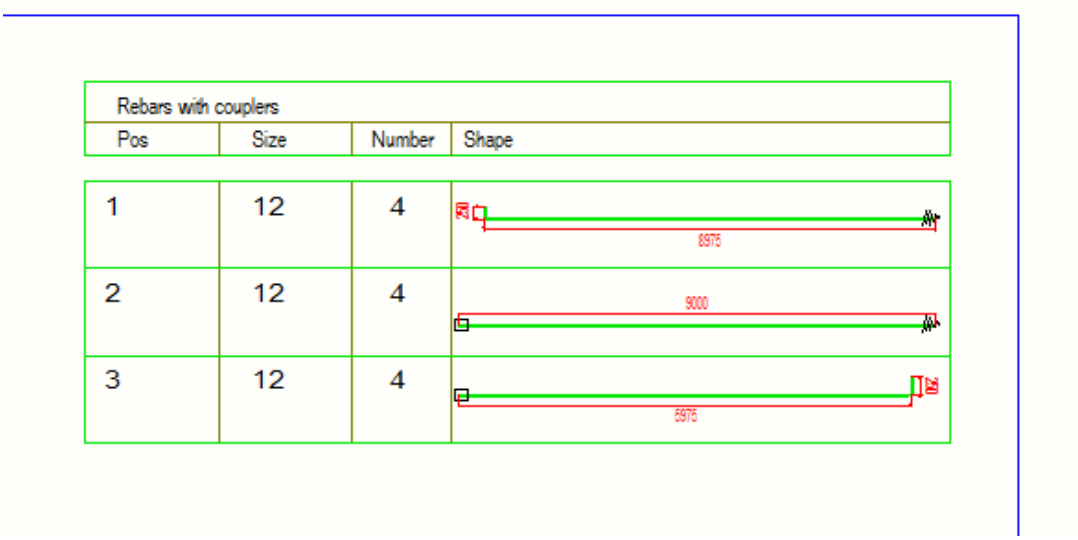
CUSTOM.REBAR_SHAPE_COUPLERS

The `CUSTOM.REBAR_SHAPE_COUPLERS` custom template attribute shows in pull-out pictures the reinforcing bar geometry, bending dimensions, and the graphical symbols representing the couplers at the bar ends. The coupler data is taken from the user-defined attributes from the rebar coupler components **Rebar coupler**, **Rebar end anchor** and **Split rebar and add coupler**.

Note that the `CUSTOM.REBAR_SHAPE_COUPLERS` attribute is available only in graphical fields when the content type is **REBAR**.

In Tekla Structures, ensure that a drawing layout contains the desired table. By default, the `rebar_with_couplers` table is available in the **Drawing layout** properties.

The drawing needs to contain at least some reinforcing bars as otherwise the table does not have anything to show.



Customize the symbols for couplers and end anchors

You can customize how the symbols for couplers and end anchors are shown.

1. You can define the mapping between the model properties and the actual symbol for various types of couplers or end anchors.

   The mapping is handled in the `RebarCoupler.Symbols.dat` file, located by default in `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\system`. The file can be placed under the model folder or under any of the common system folders defined by the advanced options `XS_PROJECT`, `XS_FIRM`, and `XS_SYSTEM`. For instructions on how to control the mapping, see the `RebarCoupler.Symbols.dat` file.

   You can give both the symbol file name and symbol number in the configuration file `RebarCoupler.Symbols.dat`. If the symbol file name is not given, the default file (`CouplerSymbols.sym`) will be used. For more details, see sample files included in the environments.

2. You can create your own symbols that are drawn at the reinforcing bar ends.

   All symbols to be used are in the symbol file `CouplerSymbols.sym`, located by default in `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\symbols`. You can create and add new symbols in Symbol Editor.

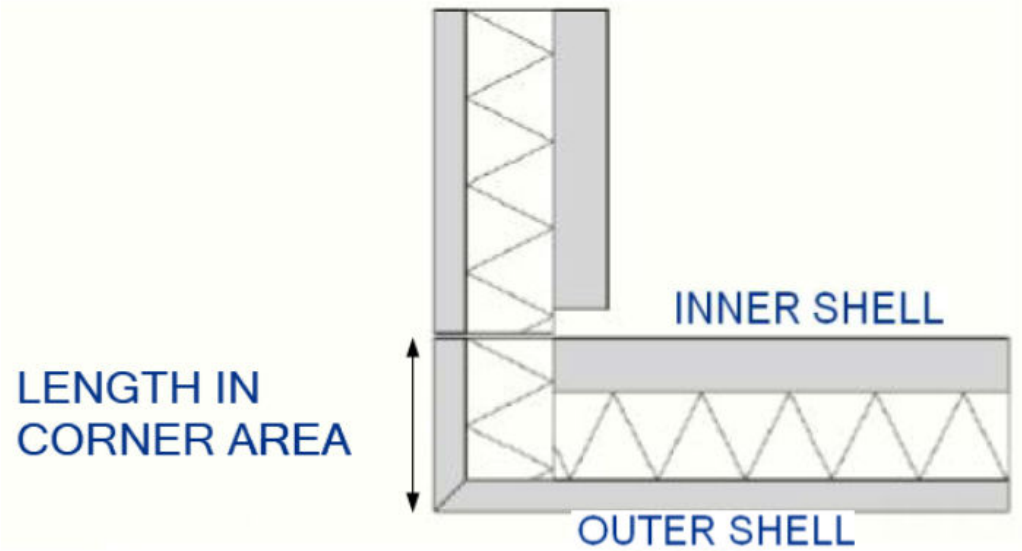
See also

Rebar coupler and anchor tools (page 3875)

CUSTOM.WALL_xxx

The following part-specific opening and area calculations are available for Sandwich wall. The calculations can be output with custom reports.

- CUSTOM.WALL_CORNER_AREA: This is the façade area of the turning corner in the wall. The turning corner part has to be at the very end of the corner to get the total length. The corner part needs to be defined in the same way as explained in the **Including turning corners in area calculation** section below.

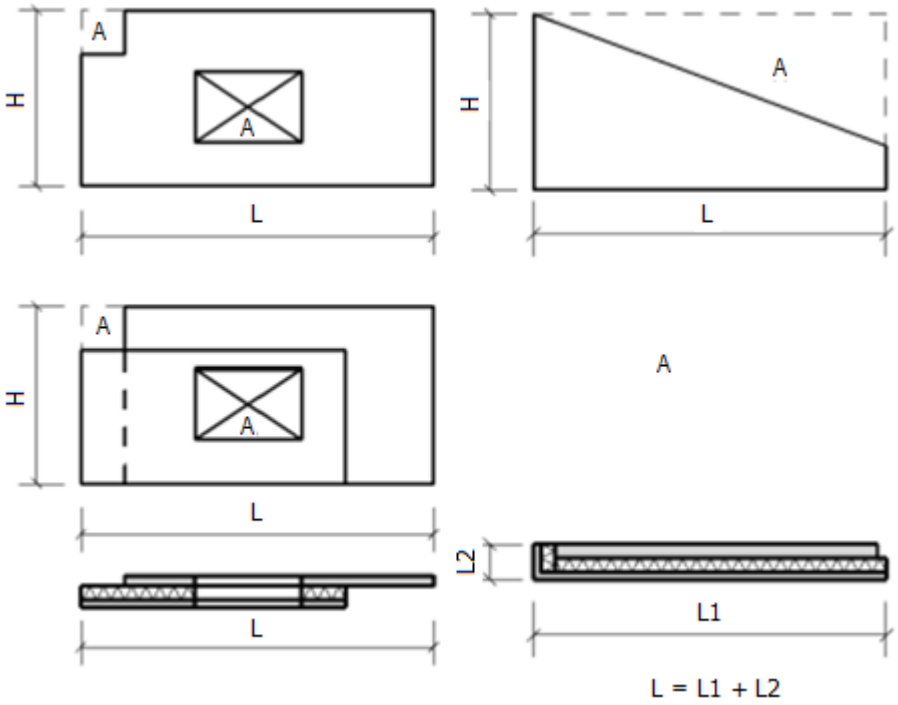


- CUSTOM.WALL_GROSS_AREA: This is the gross area of the wall.
- CUSTOM.WALL_NET_AREA: This is the net area of the wall. All openings inside the wall and/or at exterior boundaries of the wall are excluded.
- CUSTOM.WALL_OPENINGS_AREA: This is the total area of all openings inside the wall and/or at exterior boundaries of the wall.
- CUSTOM.WALL_OPENINGS_N: This is the total number of openings inside the wall and/or at exterior boundaries of the wall.

In Template Editor these attributes are located in the CUSTOM subfolder in the **Attribute** dialog box.

The examples below show the gross and net areas of sandwich walls:

- Gross area: Calculation formula: (HxL), excluding possible lifting loops or other non-concrete materials. Area of turning corner shall be included in calculation.
- Net area: Calculation formula: HxL - ΣA i



Including turning corners in area calculation

To include turning corners in area calculation, ensure that you have the name of the turning corner part (**L2** in the image above) listed in the `SandwichWallCornerPartNames.dat` file. This file lists all valid corner part names. When a report using any of these custom wall fields is generated for the first time, the file is searched in the normal file search order, starting from the model folder and then continuing the search from the folders defined for the advanced options `XS_PROJECT`, `XS_FIRM`, and `XS_SYSTEM`. The first file found will be loaded.

NOTE The `SandwichWallCornerPartNames.dat` file is not re-loaded even if another model is opened and thus it may happen that the report is based on a file from another model.

2.4 Template attributes - D

DATE

Used to be DATE. Shows the current date. If the advanced option XS_IMPERIAL_DATE (page 282) is set, the date format is mm/dd/yyyy. Otherwise the format is dd.mm.yyyy.

REVISION content type:

In drawing templates this field shows the last revision date. In REVISION lists it also shows the revision history.

DATE_APPROVED

In templates shows the approval date of the drawing entered in the **Revision Handling** dialog box.

DATE_CHECKED

This attribute shows the date when a drawing was checked. This attribute can be included in templates. The attribute field is located in the **Revision Handling** dialog box.

DATE_CREATE

Shows the creation date of the drawing. If the advanced option XS_IMPERIAL_DATE (page 282) is set, the format of the date is mm/dd/yyyy. Otherwise the format is dd.mm.yyyy.

In drawing templates this field shows the last revision date. In REVISION lists it also shows the revision history.

DATE_END

Shows the completion date of a project from the **Project properties** in **File menu** --> **Project properties**.

DATE_ISSUE

Shows the issue date of the drawing. Use with DRAWING content type.

DATE_LAST

In drawing templates this field shows the date of the last revision. In `REVISION` lists it also shows the entire revision history.

DATE_MODIFY

Shows the date of the last changes to the drawing. If the advanced option XS_IMPERIAL_DATE (page 282) is set, the date format is `mm/dd/yyyy`. Otherwise the format is `dd.mm.yyyy`.

Use in part, cast unit and assembly lists.

DATE_PLOT

Shows the date the drawing was last printed. If the advanced option XS_IMPERIAL_DATE (page 282) is set, the date format is `mm/dd/yyyy`. Otherwise the format is `dd.mm.yyyy`.

Use in drawing tables and drawing reports. You can also use this template attribute in part, assembly and cast unit lists with `DRAWING.DATE_PLOT` value field formula.

NOTE If you have set the advanced option XS_DISABLE_DRAWING_PLOT_DATE (page 165) to `TRUE`, the drawing plot date is not stored to the database. When you set it to `FALSE`, the drawing plot date is stored.

DATE_START

Shows the starting date of the project entered in the **Project properties** in **File menu** --> **Project properties**.

DELIVERY

This attribute shows the value entered in the **Delivery** box in the **Revision Handling** dialog box.

DEPTH

Shows the depth of bolt holes. The hole depth is measured from the bolt/hole reference points (yellow and magenta handles).

Use with the `HOLE` content type, for example, to report the depth of blind holes that do not extend completely through parts.

DESCRIPTION

Shows the description entered in the **Description** box in **File** --> **Project properties** .

Shows the revision **Description** entered in the **Revision Handling** dialog box for a drawing.

DESIGNER

Shows the name of the designer in the **Project properties** in **File menu** --> **Project properties**.

DesignGroup

Shows the values entered in the **Design group** box on the **Analysis** tab in the analysis properties dialog box of the part.

Analysis part properties (page 809)

DIAMETER

Shows the bolt, nut, screw, washer, stud shank, hole or part profile diameter, depending on the content type you use.

WASHER content type:

- The inner diameter of the washer.

NUT content type:

- The inner diameter of the nut.

SCREW content type:

- The screw diameter.

STUD content type:

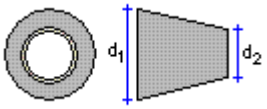
- The diameter of the stud shank.

See also

PROFILE (page 582)

DIAMETER_1, DIAMETER_2

Shows the diameters of a tapered profile. Below diameters of parametric profile PD:



See also

PROFILE (page 582)

DIAMETER_X

Shows the length of the slotted hole in the X direction (hole size + tolerance + LONG_HOLE_X).

Use with `BOLT`, `HOLE`, `NUT` and `WASHER` content types.

DIAMETER_Y

Shows the length of the slotted hole in the Y direction (hole size + tolerance + LONG_HOLE_Y).

Use with `BOLT`, `HOLE`, `NUT` and `WASHER` content types.

DIM_A ... DIM_G, DIM_H1, DIM_H2, DIM_I, DIM_J, DIM_K1, DIM_K2, DIM_L, DIM_O, DIM_R, DIM_R_ALL, DIM_TD, DIM_WEIGHT, DIM_X, DIM_Y

Show dimensions of bent reinforcing bars based on the mappings in the `rebar_schedule_config.inp` file, located in the system folder defined with

the advanced option `XS_SYSTEM`. These mappings are environment-specific by default. You can modify them to suit your company or project needs.

`DIM_TD` shows the diameter of the bending cylinder, `DIM_R` shows the radius. `DIM_R_ALL` shows multiple radiuses.

TIP When you use `DIM_R_ALL` in a value field, use `Text` as **Datatype** and `DistanceList` as **Meaning**.

See also

ANG_S, ANG_T, ANG_U, ANG_V (page 507)

DIM_A_MAX ... DIM_G_MAX, DIM_H1_MAX, DIM_H2_MAX, DIM_I_MAX, DIM_J_MAX, DIM_K1_MAX, DIM_K2_MAX, DIM_O_MAX, DIM_R_MAX, DIM_TD_MAX, DIM_X_MAX, DIM_Y_MAX

Shows the maximum dimensions of bent reinforcing bars in tapered cross sections. For more information, see .

DIM_A_MIN ... DIM_G_MIN, DIM_H1_MIN, DIM_H2_MIN, DIM_I_MIN, DIM_J_MIN, DIM_K1_MIN, DIM_K2_MIN, DIM_O_MIN, DIM_R_MIN, DIM_TD_MIN, DIM_X_MIN, DIM_Y_MIN

Shows the minimum dimensions of bent reinforcing bars in tapered cross sections. For more information, see .

DRAWING_USERFIELD_1 ... _8

Shows the value of the user-defined attribute of the drawing that you can define in the **User field 1**, **User field 2**, and so on, boxes on the **Parameters** tab in the user-defined attributes dialog box of the drawing.

DR_DEFAULT_HOLE_SIZE

Shows the default bolt hole size that you define in drawing properties. This attribute is for template purposes only.

The default bolt hole size (**Ignore size**) in the bolt mark properties defines the default size of bolt holes. This setting defines the size of bolt holes that do not have bolt marks in drawings.

DR_DEFAULT_WELD_SIZE

Defines the default weld size shown in drawing weld properties. This attribute is for template purposes only. It can be found under the **Drawing** content type in Template Editor.

The **Weld size limit** setting in drawing properties and drawing view weld properties filters welds and weld marks of the defined weld size and smaller out of the drawing.

See also

XS_WELD_FILTER_TYPE (page 500)

XS_OMITTED_WELD_TYPE (page 335)

Single-part, assembly and cast unit drawing properties (page 671)

Model weld properties in drawings (page 786)

DR_PART_POS

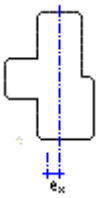
Shows the position number of the drawing main part. Can be used in drawing templates and drawing reports.

`DR_PART_POS` returns attribute `PART_POS` in all other drawing types, except for the assembly and cast unit drawings, where it returns `ASSEMBLY_POS` attribute value.

2.5 Template attributes - E

ECCENTRICITY_X, ECCENTRICITY_Y

Shows the eccentricity dimensions of a profile. Below the eccentricity x dimension of the RCXX profile:

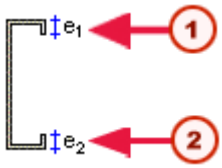


See also

PROFILE (page 582)

EDGE_FOLD, EDGE_FOLD_1, EDGE_FOLD_2

Shows the edge fold dimensions of a profile. Edge fold dimension 1 and 2 concern unsymmetrical profiles. See the example of a CC profile below:



1 EDGE_FOLD_1

2 EDGE_FOLD_2

See also

PROFILE (page 582)

END_X, END_Y, END_Z

Shows the coordinates of the end points used to create a part.

END1_ANGLE_Z

Shows the end angle of the first end of a profile in the local z-direction, for parts with cross-section profiles.

END1_ANGLE_Y

Shows the end angle of the first end of a profile in the local y-direction, for parts with cross-section profiles.

END2_ANGLE_Z

Shows the end angle of the second end of a profile in the local z-direction, for parts with cross-section profiles.

END2_ANGLE_Y

Shows the end angle of the second end of the profile in the local y-direction, for parts with cross-section profiles.

END1_CODE, END2_CODE

Shows the shape information of the first and second ends of a profile, for parts with cross-section profiles. The options are:

- 0 = no operation
- 1 = fitting
- 2 = cut
- 3 = fitting and cut

END1_SKEW, END2_SKEW

Shows 1 (INTEGER) if the corresponding end of a part has a skewed cut or fitting and 0 if the end is straight.

ERECTIONSTATUS

Shows the value selected in the **Erection Status** list on the **Status** tab in the user-defined attributes dialog box of the part.

EXTRA_LENGTH

Shows the bolt extra length.

2.6 Template attributes - F

fabricator

Shows the value entered in the **Fabricator name** box on the **Parameters** tab in the user-defined attributes dialog box of the part.

FATHER_ID

Shows the ID of the part that a reinforcement mesh belongs to.

Note that part IDs are temporary, and may change when you reopen a model, or use the read in command in Tekla Model Sharing, for example.

FINISH

Shows the final properties of a part defined in the properties dialog box (for example, in the beam properties dialog box). For all other objects the field is blank.

FLANGE_LENGTH_B

Shows the total length of the lower flange of an I profile. Use when you need to show welded profiles as plates.

FLANGE_LENGTH_U

Shows the total length of the upper flange of an I profile. Use when you need to show welded profiles as plates.

FLANGE_SLOPE_RATIO

Shows the slope ratio of a flange.

See also

PROFILE (page 582)

FLANGE_THICKNESS

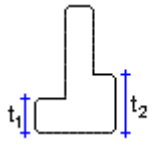
Shows the thickness of a flange.

See also

PROFILE (page 582)

FLANGE_THICKNESS_1, FLANGE_THICKNESS_2

Shows the flange thicknesses of unsymmetrical profiles, such as in unsymmetrical RCDL profile:



See also

PROFILE (page 582)

FLANGE_THICKNESS_B

Shows the thickness of the lower flange of an I profile. Use when you need to show welded profiles as plates.

See also

PROFILE (page 582)

FLANGE_THICKNESS_U

Shows the thickness of the upper flange of an I profile. Use when you need to show welded profiles as plates.

See also

PROFILE (page 582)

FLANGE_WIDTH

Shows the width of a flange.

See also

PROFILE (page 582)

FLANGE_WIDTH_1, FLANGE_WIDTH_2

Shows the flange widths of unsymmetrical profiles.

See also

PROFILE (page 582)

FLANGE_WIDTH_B

Shows the width of the lower flange of an I profile. Use when you need to show welded profiles as plates.

See also

PROFILE (page 582)

FLANGE_WIDTH_U

Shows the width of the upper flange of an I profile. Use when you need to show welded profiles as plates.

See also

PROFILE (page 582)

FOLD_ANGLE

Shows the fold angle of a profile.

See also

PROFILE (page 582)

2.7 Template attributes - G

GROUP_POS

Shows the position number of a tapered reinforcing bar group in a rebar set as defined by XS_REBARSET_TAPERED_GROUP_POSITION_NUMBER_FORMAT_STRING (page 377).

If XS_REBARSET_TAPERED_GROUP_POSITION_NUMBER_FORMAT_STRING (page 377) is not set, XS_REBAR_POSITION_NUMBER_FORMAT_STRING (page 365) defines the `GROUP_POS` format.

See also

REBAR_POS (page 586)

GROUP_TYPE

Shows the group type of a reinforcing bar:

- Normal = 0
- Tapered = 1
- Tapered 2 = 2
- Tapered curved = 3
- Tapered N = 4
- Spiral = 5

GRADE

Shows the grade of the object. Use with `BOLT`, `NUT`, `MESH`, `REBAR`, and `STUD` content types.

GUID

This template attribute shows the globally unique identifier (GUID) of an object. GUID is a permanent object property and can be used to identify objects reliably.

NOTE The report property GUID adds the prefix “ID” to the value. For example, `ID56497C3E-0000-06F6-3134-343736353635`.

2.8 Template attributes - H

HAS_CONNECTIONS

Use to check whether a part contains connections. The attribute returns 1 if the part contains connections, otherwise it returns 0.

HAS_HOLES

Use to check whether a part contains bolt holes. The attribute returns `1` if the part contains bolt holes, otherwise it returns `0`.

This attribute does not take cuts into account.

HEAD_DIAMETER

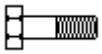
Shows the diameter of the stud head.

HEAD_THICKNESS

Shows the thickness (height) of the stud head.

HEAD_TYPE

Shows the type of the bolt head.

Bolt head type	Description	Image
1	Hex-headed	
2	Round- or cup-headed	
3	Flat-headed or countersunk	

See also

BOLT_COUNTERSUNK (page 519)

HEIGHT

Shows the height of an object.

`DRAWING` content type:

- The height of the drawing.

`ASSEMBLY` content type:

- The height of the assembly main part for assemblies, parts, and bolts.

`PART` content type:

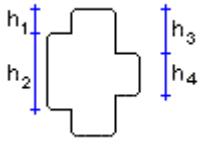
- The height of single-part or assembly drawings. Use in part and assembly lists.

See also

PROFILE (page 582)

HEIGHT_1 ... 4

Shows the height dimensions of unsymmetrical profiles, such as in RCDX profile below:



See also

PROFILE (page 582)

HIERARCHY_LEVEL

Shows the hierarchical level of an assembly. The possible values are:

- `0`: The assembly is on the highest level of the hierarchy.
- `1`: The assembly is on the highest level in a nested assembly.
- `2`: The assembly does not have any nested assemblies within it.
- Any other number: The assembly is a nested assembly within an another assembly. The number defines the level of the assembly in the assembly hierarchy.

HISTORY

Use to retrieve information on the model history. You can use this template attribute with content types `PART`, `SURFACING`, `REBAR`, `CONNECTION`, and `DRAWING`.

The following attributes can be used with the `HISTORY` attribute:

- `CREATED`
- `CREATED_BY`
- `MODIFIED`
- `MODIFIED_BY`
- `MODIFIED_ACTION`
- `TOUCHED`
- `TOUCHED_BY`
- `TOUCHED_ACTION`
- `OWNER`

Example

To find out which user has created an object in the model, use the combination `HISTORY.CREATED_BY`.

Offline usage history is stored according to the Windows domain user account. Note that in Tekla Model Sharing models, when you write out your changes to the sharing service, the changes are stored using your Trimble Identity.

Limitations

- You need to turn on the collection of model history. Set XS_COLLECT_MODEL_HISTORY (page 126) to `TRUE`.
- Information cannot be retrieved for deleted objects.
- Changes in user-defined attributes do not affect this template attribute.

HOLE.DIAMETER

The `HOLE.DIAMETER` attribute returns the diameter of the holes in drawings. It only takes account the visible holes.

HOLE_TOLERANCE

Only use in bolt lists. Shows the bolt tolerance. Shows a zero in all other lists.

HOOK_START, HOOK_END

Shows 1 if there is a hook at the start or end of a reinforcing bar, and 0 if there is no hook.

HOOK_START_ANGLE, HOOK_END_ANGLE

Shows the angle of the hook at the start or end of a reinforcing bar.

HOOK_START_LENGTH, HOOK_END_LENGTH

Shows the length of the straight part of the hook at the start or end of a reinforcing bar.

HOOK_START_RADIUS, HOOK_END_RADIUS

Shows the internal bending radius of the hook at the start or end of a reinforcing bar.

2.9 Template attributes - I

ID

Shows the identification number of an object. Use with all content types.

Note that object IDs are temporary, and may change when you reopen a model, or use the read in command in Tekla Model Sharing, for example.

IFC_BUILDING

Shows the value entered in the **IFC building name** box on the **IFC export** tab in the user-defined attributes dialog box of the part.

See also

IFC_BUILDING_STOREY

Shows the value entered in the **IFC building storey name** box on the **IFC export** tab in the user-defined attributes dialog box of the part.

See also

IFC_ENTITY

Shows the value selected in the **IFC entity** list in the **IFC export** section in the model object properties.

See also

IFC_SITE

Shows the value entered in the **IFC site name** box on the **IFC export** tab in the user-defined attributes dialog box of the part.

See also

INFO1, INFO2

Shows the corresponding values in the **Project properties** in **File menu** --> **Project properties**.

Shows the **Info 1** and **Info 2** texts of the revision entered in the **Revision Handling** dialog box.

INNER_DIAMETER

Shows the inner diameter of an object in the bolt catalog, for example, washers or nuts.

Use with BOLT, HOLE, NUT, and WASHER content types.

INSTALL_ACTUAL

Shows the value selected in the **Erection Actual** field on the **Status** tab in the user-defined attributes dialog box of the part or in the assembly properties dialog box.

INSTALL_PLAN

Shows the value selected in the **Erection Scheduled** field on the **Status** tab in the user-defined attributes dialog box of the part or in the assembly properties dialog box.

IS_BENT_PLATE

Use to check whether an object is a bent plate. You can use this attribute in filtering, for example. The attribute returns `1` if the object is a bent plate, otherwise it returns `0`.

IS_CONCEPTUAL

Use to check whether reinforcement is conceptual. The attribute returns `TRUE` if the reinforcement is conceptual, otherwise it returns `FALSE`.

IS_CURVED

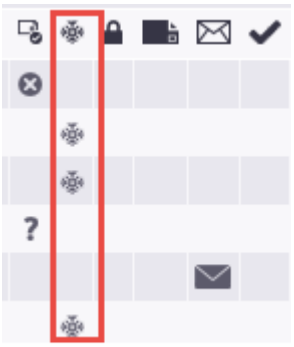
Use to check whether a reinforcing bar is curved. You can use this attribute in filtering, for example. The attribute returns `1` if the bar is curved or has a shape similar to a curved bar. Otherwise the attribute returns `0`.

IS_FROZEN

The `IS_FROZEN` attribute tells if the drawing is frozen. This attribute can be used for adding **Document manager** information about frozen drawings in drawing reports. The report returns the value 1 if the drawing is frozen, and 0 if it is not frozen.

Document manager has a column **Freeze** for this information.

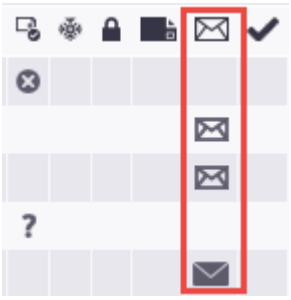
In the image below, you can see that some of the drawings are frozen (a flag in the **Freeze** column).



IS_ISSUED

The `IS_ISSUED` attribute tells if the drawing is issued. Issuing prevents the recreation of the drawing during drawing update. This attribute can be used for adding **Document manager** information about issuing in drawing reports. The report returns the value 1 if the drawing is issued, and 0 if it is not issued. Also the **Document manager** has a column **Issue** for this information.

In the image below, you can see that some of the drawings have been issued, and there is a flag in the **Issue** column. One of the issued drawings has changed, which is indicated by the flag

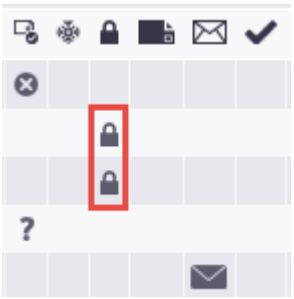


IS_ITEM

Use to check whether an object is an item. The attribute returns `1` if the object is an item, otherwise it returns `0`.

IS_LOCKED

The `IS_LOCKED` attribute tells if the drawing is locked. This attribute can be used for adding **Document manager** information about locking in drawing reports. The report returns the value 1 if the drawing is locked, and 0 if it is not locked. Also **Document manager** has a column **Lock** for this information. In the image below, you can see that two of the drawings are locked (a flag in the **Lock** column).



IS_LOFTED_PART

Use to check whether an object is a lofted plate or lofted slab. You can use this attribute in filtering, for example. The attribute returns `1` if the object is a lofted part, otherwise it returns `0`.

IS_POLYBEAM

Use to check whether a part is a polybeam. The attribute returns `1` if the part is a polybeam, otherwise it returns `0`.

IS_POUR_BREAK_VALID

Use to check whether a pour break is valid, and to find invalid pour breaks. An invalid pour break does not split a pour object completely into two. The attribute returns the value `1` if the pour break is valid and `0` if the pour break is invalid.

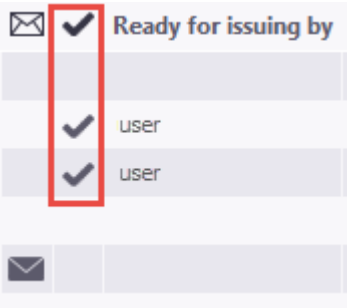
See also

XS_INVALID_POUR_BREAK_COLOR (page 289)

IS_READY_FOR_ISSUE

The `IS_READY_FOR_ISSUE` attribute tells if the drawing has been marked ready for issuing in **Document manager**. This attribute can be used for adding **Document manager** information about drawings marked for issuing in drawing reports. The report returns the value 1 if the drawing is marked ready for issuing, and 0 if it is not marked ready for issuing.

The **Document manager** has a column **Ready for issuing** for this information. If the drawing has been marked, there is a check mark in the column.



To include in the report who has marked the drawing ready for issuing, use the attribute READY_FOR_ISSUE_BY (page 585). This information is shown in the **Ready for issuing by** column in **Document manager**.

IS_REBARSET_BAR

Use to check whether a reinforcing bar belongs to a rebar set. The attribute returns `1` if the bar belongs to a rebar set, otherwise it returns `0`.

IS_SPIRAL_BEAM

Use to check whether an object is a spiral beam. You can use this attribute in filtering, for example. The attribute returns `1` if the object is a spiral beam, otherwise it returns `0`.

2.10 Template attributes - L

LAP_xxx

Use the following template attributes to show lapping information defined by using a rebar set splitter.

Template attribute	Description
`LAP_SIDE_START` `LAP_SIDE_END`	Shows the side of the lap splice from the splitter at the start or end of the bar: `Left`, `Right`, or `Middle`.
`LAP_PLACEMENT_START` `LAP_PLACEMENT_END`	Shows whether the lapping bars are parallel to each other or on top of each other at the start or end of the bar.
`LAP_LENGTH_START` `LAP_LENGTH_END`	Shows the length of the lap splice at the start or end of the bar.

See also

Splitter properties (page 652)

LAST

The last revision number of a drawing (as an integer).

LAST_APPROVED_BY

The **Approved by** information of the latest delivery of a drawing from the **Revision Handling** dialog box.

LAST_CHECKED_BY

The **Checked by** information of the latest revision from the **Revision Handling** dialog box.

LAST_CREATED_BY

The **Created by** information of the latest revision from the **Revision Handling** dialog box.

LAST_DATE_APPROVED

The approval **Date** of the latest revision of a drawing from the **Revision Handling** dialog box.

LAST_DATE_CHECKED

The **Checked by Date** of the latest revision of a drawing from the **Revision Handling** dialog box.

LAST_DATE_CREATE

In drawing templates this field shows the date of the last revision. In `REVISION` lists it also shows the entire revision history.

LAST_DELIVERY

The **Delivery** information of the latest revision from the **Revision Handling** dialog box.

LAST_DESCRIPTION

The **Description** of the latest revision from the **Revision Handling** dialog box.

LAST_INFO1

The **Info 1** text of the latest revision of the drawing from the **Revision Handling** dialog box.

LAST_INFO2

The **Info 2** text of the latest revision of the drawing from the **Revision Handling** dialog box.

LAST_MARK

In drawing templates this field shows the last revision mark. In `REVISION` lists, it also shows the entire revision history.

LAST_TEXT1...3

In drawing templates this field shows the text for the last revision. In `REVISION` lists it also shows the revision history.

LAYER

Shows the layer information of rebar set bars as defined by XS_REBARSET_REBAR_LAYER_FORMAT_STRING (page 372).

See also

LAYER_PREFIX (page 566)

LAYER_NUMBER (page 565)

LAYER_NUMBER

Shows the order number of a rebar set bar layer.

You can define layer numbers for entire rebar sets (page 639) or individual leg faces (page 643), or for individual rebar set bars using the user-defined attributes of property modifiers (page 645).

Layer numbering starts from 1. The smaller the layer number, the closer to the concrete surface the bar layer is.

See also

LAYER_PREFIX (page 566)

LAYER (page 565)

LAYER_PREFIX

Shows the prefix used for a layer of rebar set bars.

You can define the default layer prefixes of a model in the **Options** dialog box. You can also define bar layer prefixes for individual rebar sets (page 639) using their user-defined attributes, or for individual rebar set bars using the user-defined attributes of property modifiers (page 645).

See also

LAYER_NUMBER (page 565)

LAYER (page 565)

LEG_LENGTH_START, LEG_LENGTH_END

`LEG_LENGTH_START` shows the length of the first leg of the reinforcing bar. `LEG_LENGTH_END` shows the length of the last leg of the reinforcing bar.

These are the values entered in the **Start** and **End** boxes in the **Single rebar** or **Rebar group** properties (page 632) when the **Leg length** option is selected.

See also

CONCRETE_COVER_START, CONCRETE_COVER_END (page 531)

LENGTH

Shows the length of an object. Use with the following content types:

- ANALYSIS_RIGID_LINK
- ANTIMATERIAL
- ASSEMBLY
- BOLT
- CAST_UNIT
- CHAMFER
- MESH
- PART
- REBAR
- REBAR_ASSEMBLY
- SIMILAR_ASSEMBLY
- SIMILAR_CAST_UNIT
- SIMILAR_PART
- SINGLE_REBAR
- SINGLE_STRAND
- STRAND
- STUD
- WELD

Takes the cuts, fittings, and end offsets of the parts, assemblies, and cast units into account.

LENGTH_GROSS

Shows the length of assemblies, parts, and bolts before cuts are made.

LENGTH_MAX

Shows the maximum length of a reinforcing bar in a reinforcing bar group.

LENGTH_MIN

Shows the minimum length of a reinforcing bar in a reinforcing bar group.

LOCATION

Shows the location entered in the **Project properties** in **File** --> **Project properties**.

LOCKED_BY

The `LOCKED_BY` attribute tells who has locked a drawing. If the user who has locked the drawing has logged in with Trimble Identity, the account name is given, otherwise the user name is given. This attribute can be used for adding **Document manager** information about who locked the drawing in drawing reports. Also the **Document manager** has a column **Locked by** for this information. The **Lock** column has a flag when a drawing is locked.

LONG_HOLE_X

Shows the value from the **Slotted hole X** box in the bolt properties. See also DIAMETER_X (page 545).

LONG_HOLE_Y

Shows the value from the **Slotted hole Y** box in the bolt properties. See also DIAMETER_Y (page 545).

LOT_NUMBER

Shows the lot number to which the assembly belongs.

LOT_NAME

Shows the name of the lot to which the assembly belongs.

2.11 Template attributes - M

MAIN_PART

Shows `1` to indicate the main parts of assemblies and `0` for all other objects. Can be used for sorting.

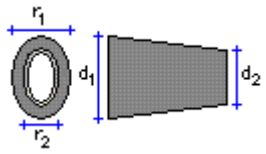
To show a main part of an assembly on top of part lists:

1. In the **Template Editor**, add value field `MAIN_PART` to `PART` row.
2. Set the **Order** to **Descending** and (if needed) hide the field in output, in the **Value Field Properties** dialog box.
3. Drag the `MAIN_PART` field to be first in sort order in the **Content browser**.

TIP To check and highlight the main part of an assembly in the model, click the down arrow next to on the ribbon, select **Assembly objects**, and then select an assembly. The main part is highlighted in orange.

MAJOR_AXIS_LENGTH_1 ... 2

Shows the major axis length dimensions of a tapered profile. Below d1 is the major axis length 1and d2 is the major axis length 2 in parametric profile EPD.



See also

PROFILE (page 582)

MARK

In drawing templates this field shows the last revision mark. In the `REVISION` lists it also shows the revision history. The revision mark of the revision entered in the **Revision Handling** dialog box.

MATERIAL

Shows the material name for parts. Shows the material of the assembly main part for assemblies. Shows the grade entered in the **Bolt assembly catalog** dialog box for bolts.

MATERIAL_TYPE

Shows the material type of assemblies or parts.

The material catalog contains the following predefined material types:

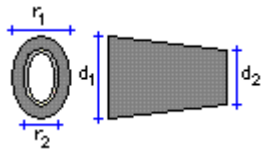
- STEEL
- CONCRETE
- TIMBER
- MISCELLANEOUS

MESH_POS

Shows the position of a mesh as defined by the advanced option XS_REBAR_POSITION_NUMBER_FORMAT_STRING (page 365).

MINOR_AXIS_LENGTH_1 ... 2

Shows the minor axis length dimensions of a tapered profile. Below r1 is the minor axis length 1and r2 is the minor axis length 2 in parametric profile EPD.



See also

PROFILE (page 582)

MODEL

Shows the name of the model.

MODEL_PATH

You can use the template attribute `MODEL_PATH` in all content types to find the path to the current model, for example `C:\TeklaStructuresModels\New Model 1\`.

MODEL_TOTAL

Shows the number of similar objects in a model (i.e. those with the same position number).

MODULUS_OF_ELASTICITY

Shows the modulus of elasticity of a material from the material catalog.

MOMENT_OF_INERTIA_X

Shows the moment of inertia around the x-x reference axis of a cross section. Moment of inertia is also known as the second moment of area.

See also

PROFILE (page 582)

MOMENT_OF_INERTIA_Y

Shows the moment of inertia around the y-y reference axis of a cross section. Moment of inertia is also known as the second moment of area.

See also

PROFILE (page 582)

moment1, moment2

Shows the values entered in the **Moment, M** box on the **End codes** tab in the user-defined attributes dialog box of the part. `moment1` shows the value in the **Start** box and `moment2` in the **End** box.

MORTAR_VOLUME

Shows the mortar volume used in surface treatment.

2.12 Template attributes - N

NAME

The name of the object. If the object does not have a name, it is searched from the next level.

Depending on the content type, shows:

Content type	Description
ASSEMBLY	The assembly mainpart, project, phase or drawing name.
BOLT	Bolt name from the bolt catalog. Nut, washer, phase, or project name.
CAST UNIT	Project, mainpart, phase or drawing name
CONNECTION	The connection name that appears in the title bar of the corresponding connection properties dialog box. Project name.
DRAWING	The entire drawing name, including the drawing type (A, W, C, G, M). Project name.
HOLE	Bolt, nut, washer, phase, or project name.
MESH	The mesh name, or project name.
NUT	The nut name, or bolt, washer, project or phase name.
PART	Name entered in the part properties dialog box for parts. Phase, assembly main part, drawing or project name.
REBAR	The reinforcing bar name. Phase or project name.

Content type	Description
STUD	The stud name. Project or phase name.
SURFACING	Surface treatment name defined in the `product_finishes.dat` file Project name.
WASHER	The washer name from the bolt catalog. Bolt, nut, project or phase name.

NAME_BASE

Shows the drawing name.

NEUTRAL_AXIS_LOCATION_ELASTIC_X

Shows the location of elastic neutral axis.

See also

PROFILE (page 582)

NEUTRAL_AXIS_LOCATION_ELASTIC_Y

Shows the location of elastic neutral axis.

See also

PROFILE (page 582)

NEUTRAL_AXIS_LOCATION_PLASTIC_X

Shows the location of plastic neutral axis.

See also

PROFILE (page 582)

NEUTRAL_AXIS_LOCATION_PLASTIC_Y

Shows the location of plastic neutral axis.

See also

PROFILE (page 582)

NORMALIZED_WARPING_CONSTANT

Shows the warping constant of a profile.

See also

PROFILE (page 582)

NUMBER, NUMBER#1, NUMBER #2

`NUMBER` shows the revision number in **Rev. No** box in the **Revision Handling** dialog box.

`NUMBER#1` shows the total number of objects on a list. Shows the total number of parts and bolts for one assembly for list types `ASSEMBLY`, `ASSEMBLY_BOLT`, `ASSEMBLY_PART` and `ASSEMBLY_ALL`, if the object is part of an assembly on the list.

`NUMBER#2` shows the project number as text.

NUMBER_IN_DRAWING

The attribute `NUMBER_IN_DRAWING` shows the number of all rebars in a drawing that have the same position number. The neighbor part rebars are ignored. Therefore, use `NUMBER_IN_DRAWING` in main reinforcement marks only, and not in marks of the visible neighbor reinforcement.

If you include the attribute `NUMBER_IN_DRAWING` in a reinforcement mark element, you get number of all rebars in the drawing that have the same position number. For meshes, `NUMBER_IN_DRAWING` returns the number of similar meshes.

Note that `NUMBER_IN_DRAWING` works in associative notes and reinforcement marks, but it does not work in filtering.

See also

NUMBER_VISIBLE (page 575)

NUMBER_IN_PHASE(X)

Returns the quantity of assemblies in phase X. The result is the same as the `NUMBER` template attribute but by phase.

You can also use the template attribute `PHASE` and the function `GetValue` instead of a number in the template attribute.

Example

```
GetValue("NUMBER_IN_PHASE(GetValue("PHASE"))")
```

NUMBER_OF_BARS_IN_GROUP

Shows the number of the reinforcing bars in a bar group.

Use with the row content type `SINGLE_REBAR` as follows:

```
REBAR.NUMBER_OF_BARS_IN_GROUP
```

See also

WEIGHT_TOTAL_IN_GROUP (page 611)

NUMBER_OF_TILE_TYPES

Returns the number of tiles in a tile pattern. For example, the **Basketweave** pattern is made up of eight tiles, so the template attribute returns `8` for a tile surface treatment whose pattern type is **Basketweave**.

NUMBER_VISIBLE

When added in the reinforcing bar group mark, shows the number of visible reinforcing bars in the view. This is a context-specific template attribute.

2.13 Template attributes - O

OBJECT

Shows project information entered in the **Object** box in **File menu** --> **Project properties**.

OBJECT_DESCRIPTION

Shows the object type and ID. Below examples:

- PART 780*380 Id: 227
- ASSEMBLY Id: 144
- MESH Id: 946

Note that object IDs are temporary, and may change when you reopen a model, or use the read in command in Tekla Model Sharing, for example.

OBJECT_LOCKED

Shows the status of the user-defined attribute **Locked**.

See also

ASSEMBLY.OBJECT_LOCKED (page 512)

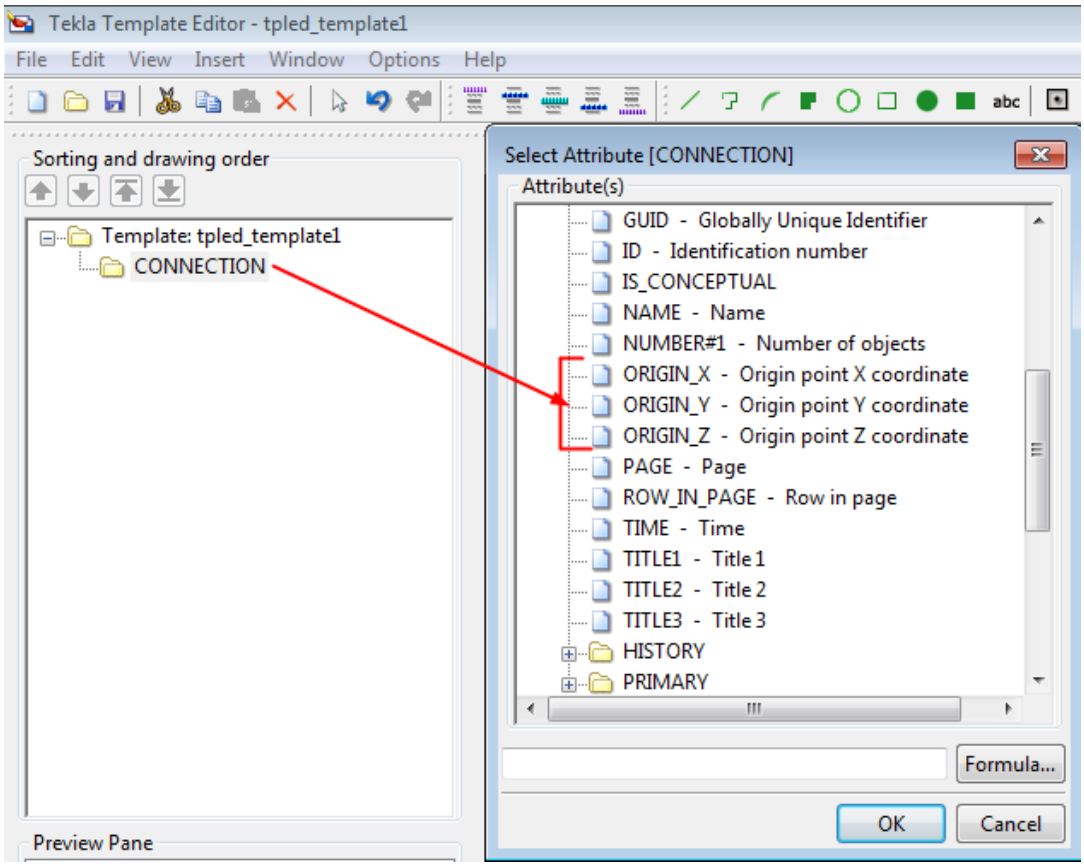
ASSEMBLY.OWNER_ORGANIZATION (page 513)

ASSEMBLY.LOCK_PERMISSION (page 512)

XS_OBJECTLOCK_DEFAULT (page 332)

ORIGIN_X, ORIGIN_Y, ORIGIN_Z

It is possible to inquire the global coordinates of a connection's origin through templates. The fields are named `ORIGIN_X`, `ORIGIN_Y` and `ORIGIN_Z`.



OBJECT_TYPE

The type of object. The message files contain the translations of these strings (numbers 576 - 587).

The object types are:

- POINT
- PART
- JOINT
- FITTING
- SCREW
- ANTI-MATERIAL
- CUT

- WELDING
- ASSEMBLY
- DRAWING
- PROJECT
- OBJECT

OWNER

For native Tekla Structures objects, shows the object owner in format `domain \user`.

2.14 Template attributes - P

PAGE

The current page number.

PART_POS

The position number of parts. Shows a blank cell for all other objects.

Shows the mark of assembly main part for assemblies, parts, and bolts. For all other objects the field is blank.

PART_PREFIX

Shows the part prefix, defined in the part properties.

For more information on numbering series, see .

PART_SERIAL_NUMBER

Shows the part number without the prefix and separator.

PART_START_NUMBER

Shows the part start number.

See also

PCS

Shows the number of bars in a reinforcing bar group.

PERIMETER

The template attribute `PERIMETER` gives the perimeter of concrete slabs or polygon plates. In **Template Editor**, the content type of this template attribute is `PART.PERIMETER`. It can be used both in textual templates and in graphical templates.

`PERIMETER` can also be used for calculating formwork area, and for take-offs.

PHASE

The number of the phase to which the object belongs.

To show the phase name, use the PHASE.NAME field.

PLASTIC_MODULUS_X

Shows the plastic modulus of x-x reference axis of a cross section. Also known as the first moment of area.

See also

PROFILE (page 582)

PLASTIC_MODULUS_Y

Shows the plastic modulus of y-y reference axis of a cross section. Also known as the first moment of area.

See also

PROFILE (page 582)

PLATE_DENSITY

Shows the material density of a plate (kg/m3).

PLATE_THICKNESS

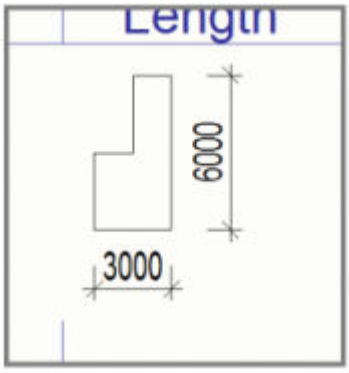
Shows the thickness of a plate (mm) if the profile has the **Plate thickness** property defined in the profile catalog. For example, it works for circular and rectangular hollow sections, and for some CC profiles which do not have separate thicknesses for flanges and web. This attribute does not work for plate profiles, because there is no **Plate thickness** that you can define in profile properties.

See also

PROFILE (page 582)

PLATE_TOP_VIEW

This attribute is available for rows with the content type **PART**. It displays the top view of the contour plate. You can use `PLATE_TOP_VIEW` in a graphical field only.



For more information about using `PLATE_TOP_VIEW`, see support article How to use PLATE_TOP_VIEW template attribute.

For more information about attributes that you can use in graphical fields, see support article Attributes for graphical fields in Template Editor.

Limitations

The following attributes have no impact on `PLATE_TOP_VIEW`: `FontColor`, `Dimensions`, `ImageWidth`, and `ImageHeight`.

PLOTFILE

Shows the name of the drawing dg file. Only for use in drawing tables and drawing reports.

See also

XS_DRAWING_PLOT_FILE_NAME_A (page 211)

XS_DRAWING_PLOT_FILE_NAME_C (page 215)

XS_DRAWING_PLOT_FILE_NAME_W (page 212)

XS_DRAWING_PLOT_FILE_NAME_G (page 213)

XS_DRAWING_PLOT_FILE_NAME_M (page 214)

POISSONS_RATIO

Shows the Poisson's ratio (analysis property) of material.

POLAR_RADIUS_OF_GYRATION

Shows the polar radius of gyration (analysis property) of a profile.

See also

PROFILE (page 582)

POSTAL_BOX

Shows the postal box entered in the **Project properties** in **File** --> **Project properties**.

POSTAL_CODE

Shows the postal code entered in the **Project properties** in **File** --> **Project properties**.

PRELIM_MARK

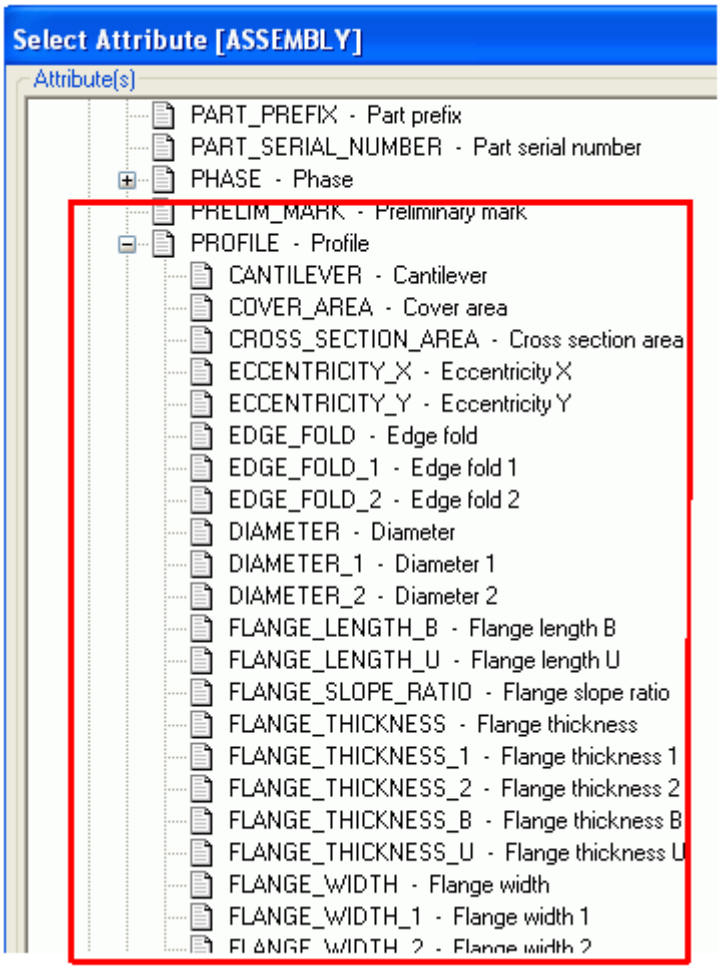
Shows the user-defined attribute **Preliminary mark**.

For more information on preliminary numbers, see .

PROFILE

Shows the part profile name, or the main part profile name in an assembly or cast unit. You can also show profile attributes in templates and reports:

In the Template editor, open the **Select attribute** dialog box and select PROFILE.* attributes for the selected content type:



For example, MAINPART.PROFILE.HEIGHT with ASSEMBLY content type, shows the height of the main part profile in the assembly.

PROFILE_DENSITY

Shows the profile density of material (kg/m3).

PROFILE_TYPE

The profile type of the part. The default types in Tekla Structures conform to DSTV-NC documentation. They are defined (message numbers 588 - 599) in the message file `by_number.ail` in the folder `..\Tekla Structures \<version>\messages`. The table below shows the relationship between messages, profiles in Tekla Structures, and the DSTV-NC profile types defined in messages.

Tekla Structures profiles	Message number	Shows DSTV-NC profile type
I-profiles	588	I
L-profiles	589	L
U-profiles	591	U
Plates	592	B
Round bars	593	RU
Round tubes	594	RO
Rectangular tubes	595	M
CC-profiles	596	C
T-profiles	597	T
Polygon plates	598	B
Bent plates	599	B
Z-profiles and all the other types of profile	590	Z

PROFILE_WEIGHT

The weight of a part. For profiles Tekla Structures calculates the weight using the weight per unit length and weight/m values in the profile catalog. If the weight/m is not defined in the profile catalog, this field works in the same way as WEIGHT_NET (page 609), but uses the plate density value (Property weight for plates) from the material catalog instead of profile density.

PROFILE_WEIGHT_NET

The net weight of a part. For profiles Tekla Structures calculates the weight using the length and weight/m values in the profile catalog. Line cuts do not affect the length value, which is calculated using the fitted centerline. For all other objects this field works in the same way as WEIGHT_NET (page 609).

PROJECT_COMMENT

Shows the value entered in the **Project Comment** box in the user-defined attributes dialog box of the project in **File menu** --> **Project properties** --> **User-defined attributes**.

PROJECT_USERFIELD_1 ... 8

Shows the value of the user-defined attribute of the project that you can define in the **User field 1** , **User field 2** and so on boxes on the **Parameters** tab in the user-defined attributes dialog box of the project (**File menu** --> **Project properties** --> **User-defined attributes**).

2.15 Template attributes - R

RADIUS

The **Radius** value of a curved beam.

RADIUS_OF_GYRATION_X

Shows the radius of gyration x (analysis property) of a profile.

See also

PROFILE (page 582)

RADIUS_OF_GYRATION_Y

Shows the radius of gyration y (analysis property) of a profile.

See also

PROFILE (page 582)

READY_FOR_ISSUE_BY

The `READY_FOR_ISSUE_BY` attribute tells who has marked a drawing ready for issuing. This attribute can be used for adding **Document manager** information about who marked the drawing ready for issuing in drawing reports, which can be checked from the **Ready for issuing by** column.

To include in a report the information whether the drawing has been marked ready for issuing, use the attribute IS_READY_FOR_ISSUE (page 562).

REBAR_ASSEMBLY_TYPE

Shows the type of a rebar assembly, selected in the rebar assembly properties. For example, `Cage`, `Bent mesh`, `Braced girder`, or blank, which is the default value.

Use with the `REBAR_ASSEMBLY` content type.

To show the rebar assembly type for a reinforcement object that belongs to a rebar assembly, use `REBAR_ASSEMBLY.USERDEFINED.REBAR_ASSEMBLY_TYPE`. A blank value is shown for reinforcement objects that do not belong to a rebar assembly.

REBAR_MESH_LEFT_OVERHANG_CROSS

Shows the extensions of the crossing bars over the outermost longitudinal bars on the left.

See also

Custom reinforcement mesh properties (page 637)

REBAR_MESH_LEFT_OVERHANG_LONG

Shows the extensions of the longitudinal bars over the outermost crossing bars on the left.

See also

Custom reinforcement mesh properties (page 637)

REBAR_MESH_RIGHT_OVERHANG_CROSS

Shows the extensions of the crossing bars over the outermost longitudinal bars on the left.

See also

Custom reinforcement mesh properties (page 637)

REBAR_MESH_RIGHT_OVERHANG_LONG

Shows the extensions of the longitudinal bars over the outermost crossing bars on the right.

See also

Custom reinforcement mesh properties (page 637)

REBAR_POS

Shows the position number of a reinforcing bar as defined by XS_REBAR_POSITION_NUMBER_FORMAT_STRING (page 365).

For reinforcing bars in tapered bar groups in rebar sets, the `REBAR_POS` format is defined by XS_REBARSET_TAPERED_REBAR_POSITION_NUMBER_FORMAT_STRING (page 379), and if that is not set, then by XS_REBAR_POSITION_NUMBER_FORMAT_STRING (page 365).

See also

GROUP_POS (page 553)

REFERENCE_ASSEMBLY

Lists assembly level information on reference models in reports and templates.

The following attributes are bound to the content type in `contentattributes_global.lst`:

```
// ---------------------
// REFERENCE_ASSEMBLY - reference model assembly
// ---------------------
REFERENCE_ASSEMBLY = NAME
REFERENCE_ASSEMBLY = BOUNDING_BOX_MIN_X
REFERENCE_ASSEMBLY = BOUNDING_BOX_MIN_Y
REFERENCE_ASSEMBLY = BOUNDING_BOX_MIN_Z
REFERENCE_ASSEMBLY = BOUNDING_BOX_MAX_X
REFERENCE_ASSEMBLY = BOUNDING_BOX_MAX_Y
REFERENCE_ASSEMBLY = BOUNDING_BOX_MAX_Z

// Logical building area attributes
REFERENCE_ASSEMBLY = LOGICAL_BUILDING_AREA.ID
REFERENCE_ASSEMBLY = LOGICAL_BUILDING_AREA.NAME
REFERENCE_ASSEMBLY = LOGICAL_BUILDING_AREA.GUID
REFERENCE_ASSEMBLY = LOGICAL_BUILDING_AREA.DEFINITION_NAME
REFERENCE_ASSEMBLY = LOGICAL_BUILDING_AREA.HIERARCHY_LEVEL
REFERENCE_ASSEMBLY = LOGICAL_BUILDING_AREA.LBA_SITE
REFERENCE_ASSEMBLY = LOGICAL_BUILDING_AREA.LBA_BUILDING
REFERENCE_ASSEMBLY = LOGICAL_BUILDING_AREA.LBA_SECTION
REFERENCE_ASSEMBLY = LOGICAL_BUILDING_AREA.LBA_STOREY

// Building object types hierarchy
REFERENCE_ASSEMBLY = OBJECT_TYPES.ID
REFERENCE_ASSEMBLY = OBJECT_TYPES.NAME
REFERENCE_ASSEMBLY = OBJECT_TYPES.GUID
REFERENCE_ASSEMBLY = OBJECT_TYPES.DEFINITION_NAME
REFERENCE_ASSEMBLY = OBJECT_TYPES.HIERARCHY_LEVEL
REFERENCE_ASSEMBLY = OBJECT_TYPES.ROOT_DEFINITION_NAME
//Project attributes
REFERENCE_ASSEMBLY = PROJECT.ADDRESS
REFERENCE_ASSEMBLY = PROJECT.BUILDER
REFERENCE_ASSEMBLY = PROJECT.DATE_END
REFERENCE_ASSEMBLY = PROJECT.DATE_START
REFERENCE_ASSEMBLY = PROJECT.DESCRIPTION
REFERENCE_ASSEMBLY = PROJECT.DESIGNER
REFERENCE_ASSEMBLY = PROJECT.INFO1
REFERENCE_ASSEMBLY = PROJECT.INFO2
REFERENCE_ASSEMBLY = PROJECT.MODEL
REFERENCE_ASSEMBLY = PROJECT.NAME
REFERENCE_ASSEMBLY = PROJECT.NUMBER#2
REFERENCE_ASSEMBLY = PROJECT.OBJECT
```

The following user-defined attributes are bound to the content type in `contentattributes_userdefined.lst`:

```
REFERENCE_ASSEMBLY = USERDEFINED.subref_description
REFERENCE_ASSEMBLY = USERDEFINED.OBJECT_LOCKED
REFERENCE_ASSEMBLY = USERDEFINED.subref_info_string
REFERENCE_ASSEMBLY = USERDEFINED.subref_logical_name
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].DESIGN_CHECKED_BY
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].DESIGN_COMMENT
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].DESIGN_ASSIGNED_TO
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].DESIGN_CODE
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].PLANS_STATUS
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].DESIGN_CHECK_DATE
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].PLANNED_START_D
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].PLANNED_END_D
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].ACTUAL_START_D
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].ACTUAL_END_D
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].FABRICATION_CODE
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].DELIVERY_NUMBER
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].PACKAGE_NUMBER
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].SHIPMENT_NUMBER
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].FABRICATION_STATUS
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].PLANNED_START_F
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].PLANNED_END_F
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].ACTUAL_START_F
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].ACTUAL_END_F
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].ERECTION_CODE
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].ERECTION_COMMENT
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].ERECTION_STATUS
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].CIP_STATUS
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].PLANNED_START_E
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].PLANNED_END_E
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].ACTUAL_START_E
REFERENCE_ASSEMBLY = USERDEFINED.[Workflow].ACTUAL_END_E
```

See also

REFERENCE_MODEL

Lists reference models in reports.

REFERENCE_MODEL_OBJECT

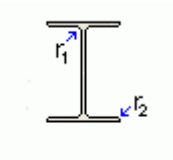
Lists reference model objects in reports.

REGION

Shows the region entered in the **Project properties** in **File** --> **Project properties**.

ROUNDING_RADIUS, ROUNDING_RADIUS_1 ... 2

Shows different rounding radii of profiles. Below as an example rounding radius 1 and 2 for a specific I profile:



See also

PROFILE (page 582)

ROW_IN_ALLPAGES

The row number incrementing continues on the next page. Use in reports and templates.

Can be used together with the `PAGE` field to include page or row information in the drawing template. Set **Type** to **Text** and enter the following field definition in the Text field properties:

```
=%PAGE% %/% %ROW_IN_ALLPAGES%
```

See also

ROW_IN_PAGE (page 589)

ROW_IN_PAGE

Produces an incremental number starting from 1 at the beginning of each new page. Use in reports and templates.

Example

Can be used together with the `PAGE` field to include page or row information in the drawing template. Set **Type** to **Text** and enter the following field definition in the Text field properties:

```
=%PAGE% %/% %ROW_IN_PAGE%
```

See also

ROW_IN_ALLPAGES (page 589)

2.16 Template attributes - S

SCALE1...5

SCALE1, SCALE2, SCALE3, SCALE4, and SCALE5 are value fields that you can use in your drawing templates.

SCALE1 shows the biggest view scale in the current drawing, SCALE2 shows the second biggest view scale, and so on. If you have more than 5 different view scales, only the 5 biggest can be displayed in your template. If you have less than 5 different view scales, the rest of these value fields will not have any value.

For example, your main view has the view scale 1:20, your detail views have the view scale 1:5, and your section views have the view scale 1:10. Now SCALE1 = "1:5" (biggest), SCALE2 = "1:10" (second biggest), SCALE3 = "1:20" (third biggest), SCALE4 = "" (empty), and SCALE5 = "" (empty).

Note that the template contents are not updated when you modify a view scale. Templates are only updated during certain operations, for example, when you reopen the drawing, or run the **Arrange Drawing Views** command.

SCHED_FAB_DATE

Shows the value selected in the **Fabrication Scheduled** field on the **Status** tab in the user-defined attributes dialog box of the part.

SCREW_HOLE_DIAMETER_X

Shows the length of a screw hole in the x direction (hole diameter + LONG_HOLE_X (page 568)).

SCREW_HOLE_DIAMETER_Y

Shows the length of a screw hole in the y direction (hole diameter + LONG_HOLE_Y (page 568)).

SECTION_MODULUS_X, SECTION_MODULUS_Y

Shows section modulus (analysis property) of a profile.

See also

PROFILE (page 582)

SHAPE

Shows the environment-specific bending type of a reinforcing bar.

SHAPE_INTERNAL

Shows the internal bending type of Tekla Structures for a reinforcing bar, for example, 2_1.

See also

SHEAR_CENTER_LOCATION

Shows the shear center location (analysis property) of a profile.

See also

PROFILE (page 582)

shear1, shear2

Shows the values entered in the **Shear, V** box on the **End codes** tab in the user-defined attributes dialog box of the part. `shear1` shows the value in the **Start** box and `shear2` in the **End** box.

SHOP_ISSUE

Shows the value selected in the **Plans Actual** field on the **Status** tab in the user-defined attributes dialog box of the part or in the assembly properties dialog box.

SHOPSTATUS

Shows the value selected in the **Fabrication Status** list on the **Status** tab in the user-defined attributes dialog box of the part or in the assembly properties dialog box.

SIMILAR_TO_MAIN_PART

Returns 1 if the position number of the given part is the same as the position number of the main part in the assembly.

To show a main part of an assembly on top of part lists:

1. In the Template Editor, add value field `SIMILAR_TO_MAIN_PART` to `PART` row.
2. Set the **Order** to **Descending** and (if needed) hide the field in output, in the **Value Field Properties** dialog box.
3. Drag the SIMILAR_TO_MAIN_PART field to be first in sort order in the **Content browser**.

SITE_WORKSHOP

For bolts this field shows the assembly type information in a string (Site or Shop). The message files (466 and 467) contain translations of these strings.

For studs this field shows the assembly type information in a string (Site or Shop).

SIZE

Shows the size of the drawing (e.g. 210x297). Only for use in drawing templates and drawing reports.

SORT_OF_E_x_Cw_PER_G_x_J

Shows sqrt(ECw/GJ) analysis property of a profile.

See also

PROFILE (page 582)

SPIRAL_ROTATION_ANGLE

Shows the total +/- angle of the rotation of a spiral beam.

For example: (+)720.00 = 2 full rounds of rotation in counterclockwise.

SPIRAL_ROTATION_AXIS_xxx

- SPIRAL_ROTATION_AXIS_BASE_POINT_X
- SPIRAL_ROTATION_AXIS_BASE_POINT_Y
- SPIRAL_ROTATION_AXIS_BASE_POINT_Z
- SPIRAL_ROTATION_AXIS_BASE_POINT_X_PROJECT
- SPIRAL_ROTATION_AXIS_BASE_POINT_Y_PROJECT
- SPIRAL_ROTATION_AXIS_BASE_POINT_Z_PROJECT
- SPIRAL_ROTATION_AXIS_BASE_POINT_X_BASEPOINT
- SPIRAL_ROTATION_AXIS_BASE_POINT_Y_BASEPOINT
- SPIRAL_ROTATION_AXIS_BASE_POINT_Z_BASEPOINT
- SPIRAL_ROTATION_AXIS_BASE_POINT_X_IN_WORK_PLANE
- SPIRAL_ROTATION_AXIS_BASE_POINT_Y_IN_WORK_PLANE
- SPIRAL_ROTATION_AXIS_BASE_POINT_Z_IN_WORK_PLANE
- SPIRAL_ROTATION_AXIS_UP_POINT_X
- SPIRAL_ROTATION_AXIS_UP_POINT_Y
- SPIRAL_ROTATION_AXIS_UP_POINT_Z
- SPIRAL_ROTATION_AXIS_UP_POINT_X_PROJECT
- SPIRAL_ROTATION_AXIS_UP_POINT_Y_PROJECT
- SPIRAL_ROTATION_AXIS_UP_POINT_Z_PROJECT
- SPIRAL_ROTATION_AXIS_UP_POINT_X_BASEPOINT
- SPIRAL_ROTATION_AXIS_UP_POINT_Y_BASEPOINT
- SPIRAL_ROTATION_AXIS_UP_POINT_Z_BASEPOINT
- SPIRAL_ROTATION_AXIS_UP_POINT_X_IN_WORK_PLANE
- SPIRAL_ROTATION_AXIS_UP_POINT_Y_IN_WORK_PLANE
- SPIRAL_ROTATION_AXIS_UP_POINT_Z_IN_WORK_PLANE

Shows the spiral beam center axis using 2 points. The rotation axis direction is calculated from those points.

SPIRAL_TOTAL_RISE

Shows the distance between the spiral beam start point and end point along the coordinate system z axis.

SPIRAL_TWIST_END

Shows the twist +/- angle of the spiral beam profile at the end of the part. The default is 0.00

SPIRAL_TWIST_START

Shows the twist +/- angle of the spiral beam profile at the start of the part. The default is 0.00

SUPPLEMENT_PART_WEIGHT

Shows the weight of supplementary parts. SUPPLEMENT_PART_WEIGHT = the weight of the whole assembly less the weight of the main part.

See also WEIGHT (page 608).

START_X

Shows the coordinates of the creation points of parts.

START_Y

See START_X (page 594).

START_Z

See START_X (page 594).

STATICAL_MOMENT_Qf

Shows the statical moment of the flange.

See also

PROFILE (page 582)

STATICAL_MOMENT_Qw

Shows the statical moment of the web.

See also

PROFILE (page 582)

STIFFENER_DIMENSION

Shows the stiffener dimension of a profile.

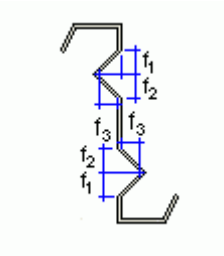
See also

STIFFENER_DIMENSION_1 ... 3 (page 595)

PROFILE (page 582)

STIFFENER_DIMENSION_1 ... 3

Shows the stiffener dimensions of a profile. Below f1 is stiffener dimension 1, f2 is stiffener dimension 2, and f3 is stiffener dimension 3 in parametric profile EZ.



See also

PROFILE (page 582)

STRAND_DEBONDED_STRANDS_1...5

Shows a list of debonded strands. Strand numbers are separated by spaces.

STRAND_DEBONDED_STRANDS_1 corresponds to row 1 on the **Debonding** tab in the **Strand Pattern Properties** dialog box, STRAND_DEBONDED_STRANDS_2 to row 2, and so on.

STRAND_DEBOND_LEN_FROM_END_1...5

Shows the debonding length from the end of the strands.

STRAND_DEBOND_LEN_FROM_END_1 corresponds to row 1 on the **Debonding** tab in the **Strand Pattern Properties** dialog box, STRAND_DEBOND_LEN_FROM_END_2 to row 2, and so on.

STRAND_DEBOND_LEN_FROM_START_1...5

Shows the debonding length from the start of the strands.

STRAND_DEBOND_LEN_FROM_START_1 corresponds to row 1 on the **Debonding** tab in the **Strand Pattern Properties** dialog box, STRAND_DEBOND_LEN_FROM_START_2 to row 2, and so on.

STRAND_DEBOND_LEN_MIDDLE_TO_END_1...5

Shows the debonding length from the middle to the end of the strands.

STRAND_DEBOND_LEN_MIDDLE_TO_END_1 corresponds to row 1 on the **Debonding** tab in the **Strand Pattern Properties** dialog box, STRAND_DEBOND_LEN_MIDDLE_TO_END_2 to row 2, and so on.

STRAND_DEBOND_LEN_MIDDLE_TO_START_1...5

Shows the debonding length from the middle to the start of the strands.

STRAND_DEBOND_LEN_MIDDLE_TO_START_1 corresponds to row 1 on the **Debonding** tab in the **Strand Pattern Properties** dialog box, STRAND_DEBOND_LEN_MIDDLE_TO_START_2 to row 2, and so on.

STRAND_N_PATTERN

Shows the number of different cross-sections in a strand pattern.

STRAND_N_STRAND

Shows the number of strands.

STRAND_POS

Shows the position (prefix and running number) of a strand.

STRAND_PULL_FORCE

Shows the pull force of a strand.

STRAND_UNBONDED

Shows the sequence numbers of debonded strands, separated by spaces or commas.

SUB_ID

Shows the running index number of a reinforcing bar in a bar group.

Use with the `SINGLE_REBAR` content type.

See also

SUB_ID_WITH_LETTERS (page 598)

SUB_ID_LAST (page 597)

SUB_ID_WITH_LETTERS_LAST (page 598)

XS_REBARSET_TAPERED_REBAR_POSITION_NUMBER_FORMAT_STRING (page 379)

SUB_ID_LAST

Shows the running index number of the last reinforcing bar in a bar group.

Use with the `REBAR` and `SINGLE_REBAR` content types.

See also

SUB_ID_WITH_LETTERS_LAST (page 598)

SUB_ID (page 597)

SUB_ID_WITH_LETTERS (page 598)

SUB_ID_WITH_LETTERS

Shows the running index number of a reinforcing bar in a rebar set bar group using letters.

Use with the `SINGLE_REBAR` content type.

See also

SUB_ID (page 597)

SUB_ID_WITH_LETTERS_LAST (page 598)

SUB_ID_LAST (page 597)

XS_REBARSET_TAPERED_REBAR_POSITION_NUMBER_FORMAT_STRING (page 379)

SUB_ID_WITH_LETTERS_LAST

Shows the running index number of the last reinforcing bar in a rebar set bar group using letters.

Use with the `REBAR` and `SINGLE_REBAR` content types.

See also

SUB_ID_LAST (page 597)

SUB_ID_WITH_LETTERS (page 598)

SUB_ID (page 597)

SUBTYPE

Shows the subtype of a profile.

See also

PROFILE (page 582)

SURFACING_NAME

Shows the name of a surface treatment, for example, Tile surface 1.

Surface treatment codes and names are defined in the `product_finishes.dat` file.

See also

CODE (page 530)

2.17 Template attributes - T

TANGENT_OF_PRINCIPAL_AXIS_ANGLE

Shows the tangent of principal axis angle (analysis property) of a profile.

See also

PROFILE (page 582)

TEXT1...3

In drawing templates this field shows the text for the last revision. In `REVISION` lists it also shows the revision history.

THERMAL_DILATATION

Shows the thermal dilatation coefficient of material.

THICKNESS

Shows the thickness of a tile in a tile pattern.

THREAD_IN_MATERIAL

Shows 1 if the thread of the screw can be inside the material to be connected and 0 if not.

TILE_NUMBER

Shows the number of tiles used in surface treatment (approximate value).

TILE_VOLUME

Shows the volume of tiles used in surface treatment, without the mortar volume. See also MORTAR_VOLUME (page 571).

TIME

Shows the current time (hh:mm:ss).

TITLE

Shows the user-defined drawing name. Can also be used for parts and assemblies. For example, you could create a report of assemblies listing which assembly drawings had been created.

TITLE1...3

In reports this field shows the user-defined titles entered in the **Report** dialog box. In drawing templates this field shows the drawing attributes.

TOP_LEVEL

Shows the top level of a single part, cast unit, assembly, part of a connection or a pour object.

Top level takes the unit and accuracy from `MarkDimensionFormat.dim`. You can modify the settings in `MarkDimensionFormat.dim` in the **Dimension Properties** dialog box in a drawing.

The datum level only affects the `TOP_LEVEL` attribute when **Location by** is set to **Model origin** or to the project base point that is in the model origin.

You can use this attribute as a user-defined attribute also in part marks and associative notes.

NOTE This attribute returns the value as text, so you cannot use formulae with this attribute. Use TOP_LEVEL_UNFORMATTED (page 601) instead.

TOP_LEVEL_GLOBAL

Shows the top level of a single part, cast unit, assembly, part of a connection or a pour object by global axis. `TOP_LEVEL_GLOBAL` takes the unit and accuracy from `MarkDimensionFormat.dim`. You can modify the settings in `MarkDimensionFormat.dim` in the **Dimension Properties** dialog box in a drawing.

You can use this attribute as a user-defined attribute in part marks and associative notes, and also in reports and templates.

TOP_LEVEL_GLOBAL_UNFORMATTED

Shows the top level of a single part, cast unit, assembly, part of a connection or a pour object. `TOP_LEVEL_GLOBAL_UNFORMATTED` returns the top levels as a length in mm so you can format them and include them into formulas in templates. This attribute gives level information by the global axis.

You can use this attribute as a user-defined attribute also in part marks and associative notes.

TOP_LEVEL_UNFORMATTED

Shows the top level of a single part, cast unit, assembly, part of a connection or a pour object. `TOP_LEVEL_UNFORMATTED` returns the top levels as a length in mm so you can format them and include them into formulas in templates.

The datum level only affects the attribute `TOP_LEVEL_UNFORMATTED` when **Location by** is set to **Model origin** or to the project base point that is in model origin.

You can use this attribute as a user-defined attribute also in part marks and associative notes.

NOTE Unlike the `TOP_LEVEL` attribute, the `TOP_LEVEL_UNFORMATTED` attribute cannot be formatted through the `MarkDimensionFormat.dim` file.

TORSIONAL_CONSTANT

Shows the torsional constant (analysis property) of a profile.

See also

PROFILE (page 582)

TOWN

Shows the city entered in the **Project properties** in **File** --> **Project properties**.

TYPE

Shows the object type or standard:

Content type	Description
ANALYSIS_RIGID_LINK	Shows whether the analysis rigid link was created automatically (`auto`), or added manually by a user (`user`).
BOLT	Shows the bolt standard as it appears in the **Bolt assembly catalog** dialog box (for example, 7968).
DRAWING	Shows the drawing type: A, W, C, G or M.

Content type	Description
MESH	Shows the mesh type: Rectangle, Polygon, or Bent.
NUT	Shows the nut standard.
SURFACE	Shows the surface type: Formwork or Concrete finish.
SURFACING	Shows the surface treatment type: Concrete finish, Special mix, Tile surface, or Steel finish.
WASHER	Shows the washer standard.

For all other objects the field is blank.

TYPE1

For bolts, shows the bolt standard and the standard of each possible washer or nut as they appear in the **Bolt assembly catalog** dialog box (for example, `7968/2041/2041/2041/2067/2067`). For objects other than bolts, the field is blank.

See also

TYPE2 (page 603)

TYPE3 (page 603)

TYPE4 (page 604)

TYPE2

For bolts, shows `1` for existing and `0` for non-existing washers and nuts (for example, `10011`). For objects other than bolts, the field is blank.

See also

TYPE1 (page 603)

TYPE3 (page 603)

TYPE4 (page 604)

TYPE3

The same as `TYPE2`, but shows `X` for existing and `o` for non-existing washers and nuts (for example, `XooXX`). For objects other than bolts, the field is blank.

See also

TYPE1 (page 603)

TYPE2 (page 603)

TYPE4 (page 604)

TYPE4

The same as `TYPE1`, but only shows the standard of the existing bolt elements. For objects other than bolts, the field is blank.

See also

TYPE1 (page 603)

TYPE2 (page 603)

TYPE3 (page 603)

2.18 Template attributes - U

USAGE

Shows if a reinforcing bar is a main bar, or a tie or stirrup. The attribute returns `Main bar` for main bars, and `Tie or stirrup` for ties and stirrups. If the type of use cannot be defined, the attribute returns a blank value.

See also

USAGE_VALUE (page 604)

USAGE_VALUE

Shows if a reinforcing bar is a main bar, or a tie or stirrup. The attribute returns `1` for main bars, and `2` for ties and stirrups. If the type of use cannot be defined, the attribute returns `0`.

See also

USAGE (page 604)

USERDEFINED.REBARSET_GROUP_GUID

Shows the globally unique identifier of a bar group in a rebar set.

Use with the `REBAR` or `SINGLE REBAR` content type, and with or without the `USERDEFINED.` prefix.

See also

USERDEFINED.REBARSET_GUID (page 605)

USERDEFINED.REBARSET_GUID

Shows the globally unique identifier of a rebar set.

Use with the `REBAR` or `SINGLE REBAR` content type, and with or without the `USERDEFINED.` prefix.

See also

USERDEFINED.REBARSET_GROUP_GUID (page 604)

USER_FIELD_1 ... _8

Shows the value of the user-defined attribute **User field 1**, **User field 2** etc.

See also

For more information on the user-defined attributes in templates and reports, see .

For more information on user-defined attributes, see and .

USER_PHASE

Shows the value entered in the **User Phase** box in the user-defined attributes of the part.

2.19 Template attributes - V

VOLUME

Shows the object volume, for example, the volume of an assembly or of a cast unit. Takes holes and cuts into account.

VOLUME_GROSS

Shows the object gross volume. Does not take into account holes and cuts.

VOLUME_NET

Shows the object volume taking into account holes and cuts.

VOLUME_NET_ONLY_CONCRETE_PARTS

Shows the volume by the solid of concrete parts in the cast unit. If a part uses a profile where cross section area is defined manually, it is ignored in calculation (cf. `VOLUME_ONLY_CONCRETE_PARTS`)

VOLUME_ONLY_CONCRETE_PARTS

This attribute gets cast unit volume values only for concrete parts. Reinforcing bars or embeds are not taken into account.

VOLUME_ONLY_POUR_OBJECT

Shows the volume of the concrete (=pour object) in a pour unit. Takes holes and cuts into account.

See also

WEIGHT_ONLY_POUR_OBJECT (page 610)

2.20 Template attributes - W

WARPING_CONSTANT

Shows the warping constant (analysis property) of a profile.

See also

PROFILE (page 582)

WARPING_STATICAL_MOMENT

Shows the warping statical moment (analysis property) of a profile.

See also

PROFILE (page 582)

WEB_HEIGHT

See WEB_WIDTH (page 607).

WEB_LENGTH

The gross length of the web of an I profile. Use to show welded profiles as plates.

WEB_THICKNESS

The thickness of the web of an I profile. Use to show welded profiles as plates.

See also

PROFILE (page 582)

WEB_THICKNESS_1, WEB_THICKNESS_2

The additional thickness values of the web of a profile.

See also

PROFILE (page 582)

WEB_WIDTH

The width of the web of an I profile. Use to show welded profiles as plates.

WEIGHT

Shows the weight of the object.

The calculation formula depends on the object type:

- For parts with cross-sections defined in the profile catalog, the weight is calculated from the cross section area in the profile catalog (on the list of **Properties** on the **Analysis** tab), length (`LENGTH`) and density of material (property weight for profiles in the material catalog). The result is the same as calculating `WEIGHT_GROSS`.
- For other profiles with no cross sections defined (typically parametric profiles), shows the net weight calculated using the profile volume and density of material. Fittings, cuts, weld preparations, and part adds affect volume calculation.
- For parts with surface treatment, shows both the weight of the part and the surface treatment.
- For assemblies, shows the sum of the part weights for each assembly.
- For reinforcement, shows the weight of one bar in the group. `WEIGHT_TOTAL` shows the weight of all bars in the group.
- For rebar assemblies, shows the sum of all objects' weights for each rebar assembly.
- For surface treatment, shows the weight of the surface treatment.
- For bolts, shows the weight of the bolt element in the corresponding content type rows:
  - `BOLT`: shows the weight of the bolt.
  - `NUT`: shows the weight of the nut.
  - `WASHER`: shows the weight of the washer.

WEIGHT_GROSS

Shows the gross weight, which is the total weight of material needed to fabricate the part. The calculation formula depends on the part:

- If the part has cross-sections defined in the profile catalog, the weight is calculated from part length (`LENGTH`), the cross section area in the profile catalog, and the density of material.

- If the part is a folded or contour plate without a cross section area, the weight is calculated from plate overall height, overall length and density of material (property weight for plates in the material catalog).
- For other profiles without cross sections (typically parametric profiles), the gross weight is calculated the same way as the `WEIGHT_NET`, but cuts are not taken into account and the plate density value is used instead of profile density.
- For assemblies, shows the combined gross weight of parts included in an assembly. For bolts it shows the bolt weight.

WEIGHT_M

Shows the property weight of a profile (defined in the material catalog). For parametric profiles, shows the weight of the profile divided by the length. For standard profiles, shows the **Weight per unit length** from the **Analysis** properties in the profile catalog.

WEIGHT_MAX

Shows the maximum weight of a single reinforcing bar or strand in a reinforcing bar group.

WEIGHT_MIN

Shows the minimum weight of a single reinforcing bar or strand in a reinforcing bar group.

WEIGHT_NET

Shows the weight of the fabricated part, assembly or cast unit. The calculation formula depends on the object:

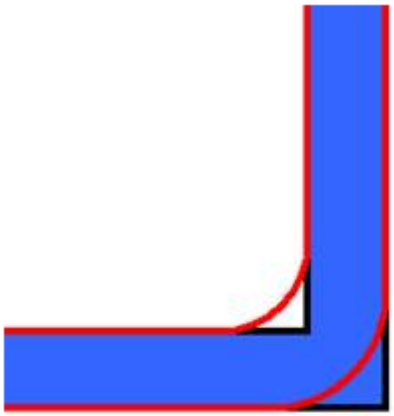
- For parts, returns the net weight, which is the actual weight of the fabricated part. Rounding of the profile corners are not taken into account.
- For bolts, returns the bolt weight, and for other objects a zero.
- For assemblies, returns the sum of part weights.

The calculation is based on part volume and density of material. The density value used in the calculation depends on the profile cross-sections:

- If cross-sections are defined in the profile catalog, density is the value of **Property: Profile Density** in the material catalog.

- If there are no cross-sections, density is the value of **Property: Plate Density** in the material catalog.

NOTE For parts, the net weight is **not** the actual weight of the fabricated parts. The profile cross section is calculated using straight angles, so the roundings in the corners are not taken into account (unless you are using the advanced option `XS_SOLID_USE_HIGHER_ACCURACY`). This causes significant difference between the calculated and the actual weight especially when big cross sections are used.



WEIGHT_NET_ONLY_CONCRETE_PARTS

Shows the weight of a cast unit. It calculates the weight by the solid of the concrete parts in the cast unit. If a part uses a profile where cross section area is defined manually, it is ignored in calculation (cf. `WEIGHT_ONLY_CONCRETE_PARTS`).

WEIGHT_ONLY_CONCRETE_PARTS

This attribute gets the cast unit weight only for concrete parts only.

WEIGHT_ONLY_POUR_OBJECT

Shows the weight of the concrete (=pour object) in a pour unit. Takes holes and cuts into account. Reinforcing bars or embeds are not taken into account.

Pour object weight is calculated according to the pour object solid and the density of the material.

See also

VOLUME_ONLY_POUR_OBJECT (page 606)

WEIGHT_ONLY_REBARS

Shows the weight of all reinforcement in a pour unit, including reinforcing bars, meshes, and strands. Does not include the weight of reinforcement belonging to precast cast units inside the pour unit.

See also

WEIGHT_ONLY_POUR_OBJECT (page 610)

WEIGHT_PER_UNIT_LENGTH

Shows the weight per unit length (analysis property) of a profile.

See also

PROFILE (page 582)

WEIGHT_TOTAL

Shows the total weight of all reinforcing bars or of all strands in a reinforcing bar group. This template attribute is available in content type `REBAR` in graphical and textual templates.

WEIGHT_TOTAL_IN_GROUP

Shows the total weight of the reinforcing bars in a bar group.

Use with the row content type `SINGLE_REBAR` as follows:

```
REBAR.WEIGHT_TOTAL_IN_GROUP
```

See also

NUMBER_OF_BARS_IN_GROUP (page 575)

WELD_ACTUAL_LENGTH1, WELD_ACTUAL_LENGTH2

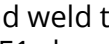
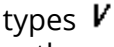
Shows the actual weld length in the model, or the sum of actual weld lengths, for welds above and below line.

The actual weld length is the distance between the weld seam start point and end point along the weld seam.

See also

WELD_LENGTH1 ... 2 (page 615)

WELD_ADDITIONAL_SIZE1, WELD_ADDITIONAL_SIZE2

Use these attributes to show the additional weld size. The additional weld size can be set for compound weld types  and  .
WELD_ADDITIONAL_SIZE1 shows the additional size value for the welds above line, and WELD_ADDITIONAL_SIZE2 for the welds below line.

These attributes can be used in report templates.

WELD_ANGLE1, WELD_ANGLE2

Shows weld angle for welds above and below line.

WELD_ASSEMBLYTYPE

Shows the assembly type of a weld (Site or Workshop). Only use in welding lists.

WELD_DEFAULT

Shows the default weld size according to the drawing attributes. Only use in drawing tables.

WELD_CROSSSECTION_AREA1, WELD_CROSSSECTION_AREA2

Shows the theoretical cross section area for supported solid weld objects above and below line. For unsupported weld types, shows 0.00.

WELD_EDGE_AROUND

Returns the value selected in the **Edge/Around** list in the **Weld** properties: **Edge** if only one edge of a face is welded and **Around** if the entire perimeter is welded.

WELD_EFFECTIVE_THROAT, WELD_EFFECTIVE_THROAT2

Use these attributes to show the weld effective throat.
`WELD_EFFECTIVE_THROAT` shows the value for the welds above line, and `WELD_EFFECTIVE_THROAT2` for the welds below line.

WELD_ELECTRODE_CLASSIFICATION

Shows the weld electrode classification, selected in the **Electrode classification** list in the **Weld** properties.

WELD_ELECTRODE_COEFFICIENT

Shows the value entered in the **Electrode coefficient** box in the **Weld** properties.

WELD_ELECTRODE_STRENGTH

Shows the value entered in the **Electrode strength** box in the **Weld** properties.

WELD_ERRORLIST

Shows error codes for a weld if there are issues related to the weld.

The error codes are:

Error code	Description
E1	Weld is not in the correct location.
E2	Welded parts are not touching each other.
E3	Weld is not on the edge of a part.
E4	Weld has a cross section type that is not supported.
E5	Weld properties are incorrect.
E6	There are issues related to the weld preparation of parts.
E7	Weld seams are far apart from each other.

WELD_FATHER_CODE

Shows the connection running number of the connection where the weld is located. The field is blank if the weld is not next to a connection. Use only in welding lists.

WELD_FATHER_NUMBER

Shows the connection number of the connection where the weld is located. The field is blank if the weld is not next to a connection. Use only in welding lists.

WELD_FILLTYPE1, WELD_FILLTYPE2

Shows the weld contour (None, Flush, Convex, Concave) for welds above and below line.

WELD_FINISH1, WELD_FINISH2

Shows the weld finish for welds above and below line.

WELD_INCREMENT_AMOUNT1, WELD_INCREMENT_AMOUNT2

Shows the amount of increments for intermittent welds above and below line.

WELD_INTERMITTENT_TYPE

Shows the shape of a weld (Continuous, Chain intermittent, or Staggered intermittent).

WELD_LENGTH1 ... 2

Shows the weld length value entered in the **Length** box in the weld properties. `WELD_LENGTH1` shows the length of the weld above the line and `WELD_LENGTH2` below the line.

See also

WELD_ACTUAL_LENGTH1, WELD_ACTUAL_LENGTH2 (page 611)

WELD_NDT_INSPECTION

Shows the non-destructive testing and inspection level of a weld, selected in the **NDT inspection** list in the **Weld** properties.

WELD_NUMBER

Shows the weld number.

See also

Weld numbering settings (page 630)

WELD_PERIOD1 ... 2

Shows the value entered in the **Pitch** box in the weld properties.
`WELD_PERIOD1` shows the value for the welds above line, and `WELD_PERIOD2` for the welds below line.

WELD_POSITION

Shows the weld position, selected in the **Position** list in the **Weld** properties.

WELD_POSITION_X

Shows the position of the weld in the x axis.

WELD_POSITION_Y

Shows the position of the weld in the y axis.

WELD_POSITION_Z

Shows the position of the weld in the z axis.

WELD_PROCESS_TYPE

Shows the welding process type of a weld, selected in the **Process type** list in the **Weld** properties.

WELD_ROOT_FACE_THICKNESS, WELD_ROOT_FACE_THICKNESS2

Shows the root face thickness of a weld above or below line and is used only in welding lists.

WELD_ROOT_OPENING, WELD_ROOT_OPENING2

Shows the root opening (space between the welded parts) for welds above and below line.

WELD_SIZE1, WELD_SIZE2

Shows weld size for welds above and below line.

WELD_SIZE_PREFIX_ABOVE

Shows the weld size prefix, entered in the **Prefix** box in the **Above line** section in the **Weld** properties.

WELD_SIZE_PREFIX_BELOW

Shows the weld size prefix, entered in the **Prefix** box in the **Below line** section in the **Weld** properties.

WELD_TEXT

Shows the reference text of a weld.

The maximum number of characters that can be shown is 80, including one character for each line of text. To show long reference texts in reports, also adjust the template field length accordingly.

WELD_TYPE1, WELD_TYPE2

Shows weld type for above and below line. See the .

WELD_VOLUME

Shows the volume of a solid weld object. If the solid weld object fails, shows 0.00. For unsupported weld types, shows 0.00.

WIDTH

The width of a part or assembly.

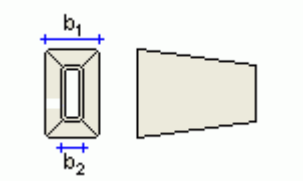
For drawings, shows the width of the drawing.

See also

PROFILE (page 582)

WIDTH_1, WIDTH_2

Shows special width values of some profiles. Below parametric profile rectangular hollow section with subtype h1*b1-h2*b2*t, where b1 is width 1 and b2 is width 2.



See also

PROFILE (page 582)

2.21 Template attributes - X

xs_shorten

Shows the value entered in the **Shorten** box on the **Parameters** tab in the user-defined attributes dialog box of the part.

3 Settings reference

This section contains detailed information on different settings.

TIP In most parts of the user interface, you can press F1 on your keyboard to view information related to the part of the user interface you are viewing. In the ribbon, hover the mouse pointer over the tool to view a tooltip and press Ctrl+F1 when the tooltip is displayed to see more information about the tool.

Use the table of contents to browse through these reference pages.

3.1 Modeling settings

This section provides more information about some of the settings you can modify in Tekla Structures.

Click the links below to find out more:

- Part position settings (page 620)
- Numbering settings (page 629)
- Reinforcement settings (page 632)

Part position settings

This section provides more information about specific part position settings. These settings can be modified in the **Position** and **End offset** sections in the part property pane, or by using the contextual toolbar.

Click the links below to find out more:

Part position on the work plane (page 621)

Part rotation (page 622)

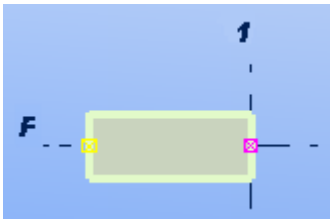
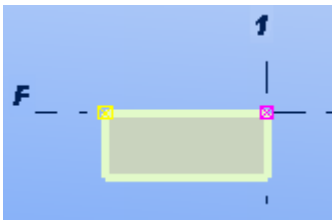
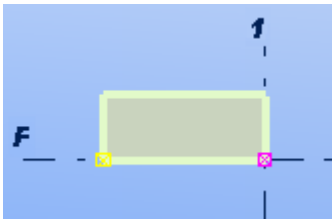
Part position depth (page 623)

Part vertical position (page 624)
Part horizontal position (page 626)
Part end offsets (page 627)

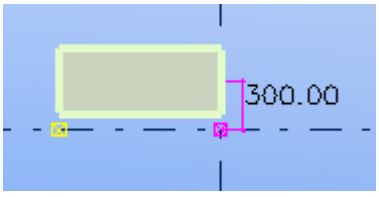
Part position on the work plane

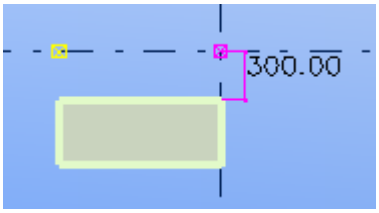
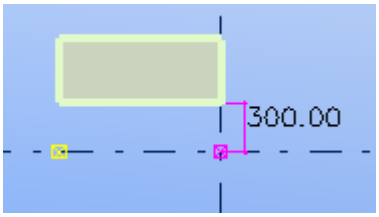
Use the **On plane** setting in the part properties to view and change the part's position on the work plane. The position is always relative to part's reference line.

Alternatively, you can use the contextual toolbar or keyboard shortcuts to modify the part's position.

Option	Description	Example
Middle	The reference line is in the middle of the part.	
Right	The part is positioned underneath the reference line.	
Left	The part is positioned above the reference line.	

Examples

Position	Example
Middle 300	

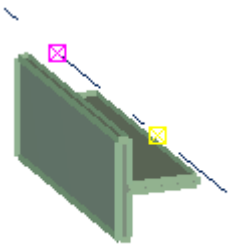
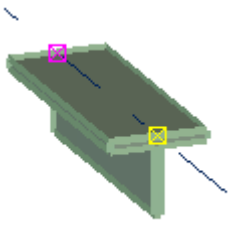
Position	Example
Right 300	
Left 300	

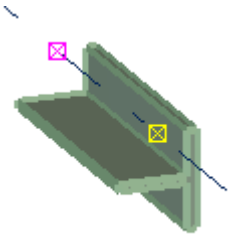
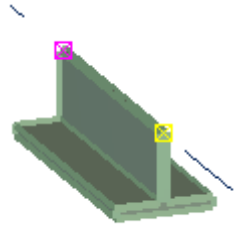
Part rotation

Use the **Rotation** setting in the part properties to view and change the rotation of a part around its axis on the work plane.

You can also define the angle of rotation. Tekla Structures measures positive values clockwise around the local x axis.

Alternatively, you can use the contextual toolbar or keyboard shortcuts to modify the part's position.

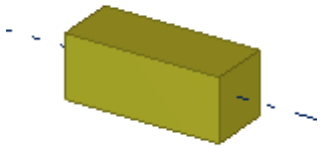
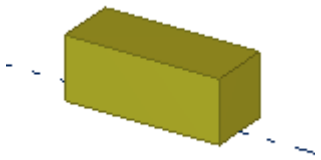
Option	Description	Example
Front	The work plane is parallel to the front plane of the part.	
Top	The work plane is parallel to the top plane of the part.	

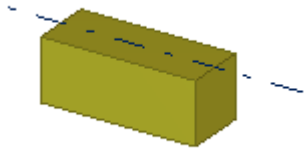
Option	Description	Example
Back	The work plane is parallel to the back plane of the part.	
Below	The work plane is parallel to the bottom plane of the part.	

Part position depth

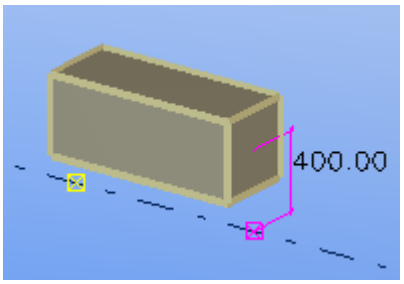
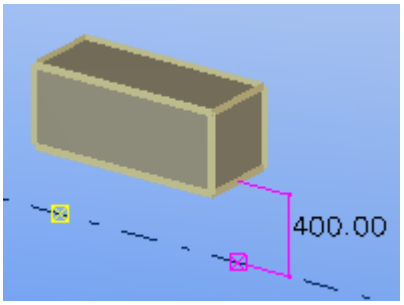
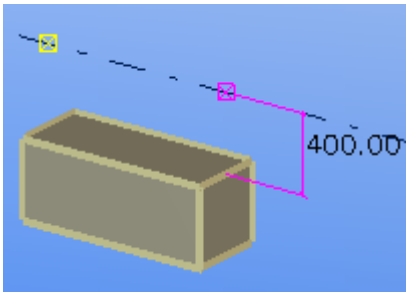
Use the **At depth** setting in the part properties to view and change the position depth of the part. The position is always relative to the part reference line between the part handles.

Alternatively, you can use the contextual toolbar or keyboard shortcuts to modify the part's position.

Option	Description	Example
Middle	The part is positioned in the middle of the reference line.	
Front	The part is positioned above the reference line.	

Option	Description	Example
Behind	The part is positioned underneath the reference line.	

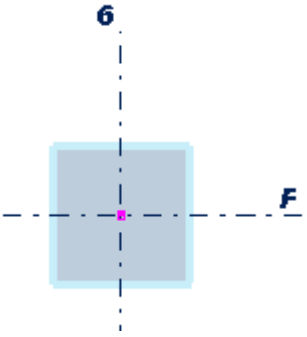
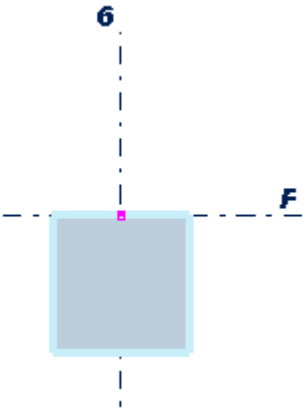
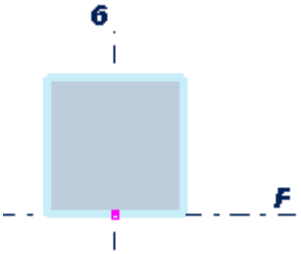
Examples

Position	Example
Middle 400	
Front 400	
Behind 400	

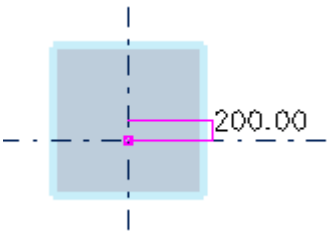
Part vertical position

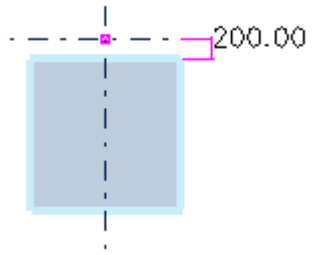
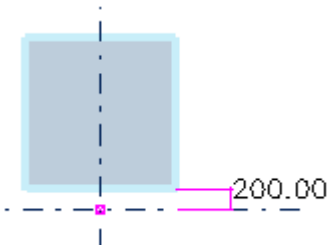
Use the **Vertical** setting in the part properties to view and change the vertical position of the part. The position is always relative to the part's reference point.

Alternatively, you can use the contextual toolbar to modify the part's position.

Option	Description	Example
Middle	The reference point is in the middle of the part.	
Down	The part is positioned underneath the reference point.	
Up	The part is positioned above the reference point.	

Examples

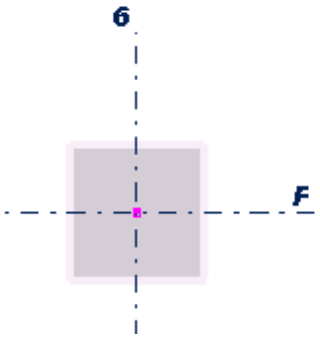
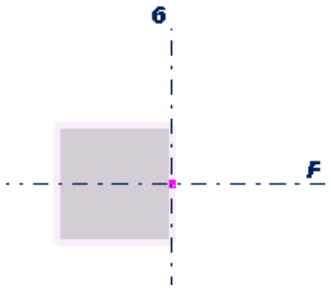
Position	Example
Middle 200	

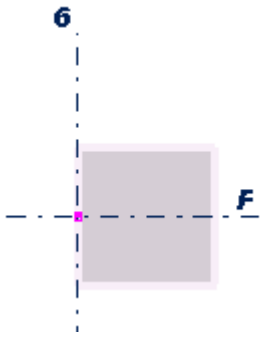
Position	Example
Down 200	
Up 200	

Part horizontal position

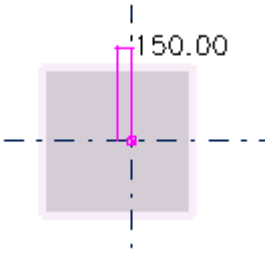
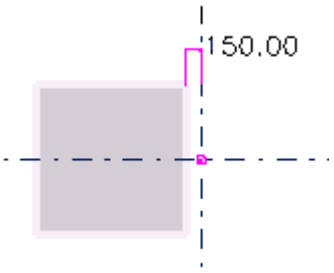
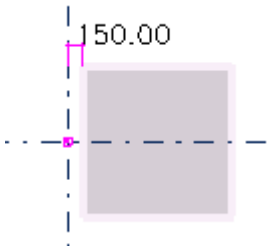
Use the **Horizontal** setting in the part properties to view and change the horizontal position of the part. The position is always relative to the part's reference point.

Alternatively, you can use the contextual toolbar to modify the part's position.

Option	Description	Example
Middle	The reference point is in the middle of the part.	
Left	The part is positioned on the left side of the reference point.	

Option	Description	Example
Right	The part is positioned on the right side of the reference point.	

Examples

Position	Example
Middle 150	
Left 150	
Right 150	

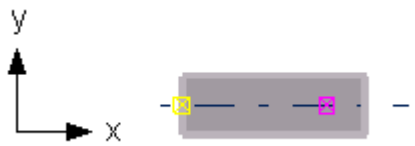
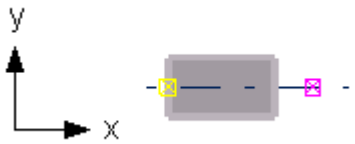
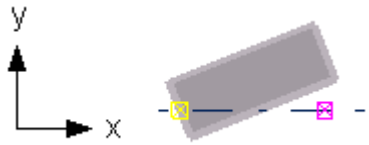
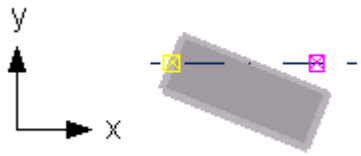
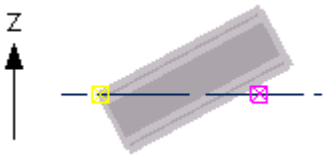
Part end offsets

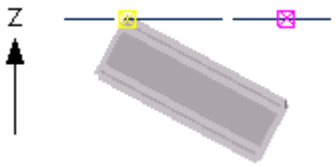
Use the **Dx**, **Dy** and **Dz** settings in the part properties to move the ends of a part, relative to its reference line. You can enter positive and negative values.

Alternatively, you can use the contextual toolbar to modify the part's position.

Option	Description
Dx	Changes the length of the part by moving the part end point along the reference line.
Dy	Moves the part end perpendicular to the reference line.
Dz	Moves the part end in the z direction of the work plane.

Examples

Position	Example
Dx End point: 200	
Dx End point: -200	
Dy End point: 300	
Dy End point: -300	
Dz End point: 400	

Position	Example
Dz End point: -400	

Numbering settings

This section provides more information about specific numbering settings.

Click the links below to find out more:

- General numbering settings (page 629)
- Weld numbering settings (page 630)
- Control number settings (page 631)

General numbering settings

Use the **Numbering Setup** dialog box to view and modify some general numbering settings.

Setting	Description
Renumber all	All parts get a new number. All information on previous numbers is lost.
Re-use old numbers	Tekla Structures reuses the numbers of parts that have been deleted. These numbers may be used to number new or modified parts.
Check for standard parts	If a separate standard-part model has been set up, Tekla Structures compares the parts in the current model to those in the standard-part model. If the part to be numbered is identical to a part in the standard-part model, Tekla Structures uses the same part number as in the standard-part model.
Compare to old	The part gets the same number as a previously numbered similar part.
Take new number	The part gets a new number even if a similar numbered part already exists.
Keep number if possible	Modified parts maintain their previous numbers if possible. Even if a part or assembly becomes identical

Setting	Description
	with another part or assembly, the original position number is maintained. For example, you might have two different assemblies, B/1 and B/2, in the model. Later on you modify B/2 so that it becomes identical with B/1. If the **Keep number if possible** option is used, B/2 will maintain its original position number when you renumber the model.
Synchronize with master model (save-numbering-save)	Use this setting when working in multi-user mode. Tekla Structures locks the master model and performs a save, numbering, and save sequence, so that all other users can continue working during the operation.
Automatic cloning	If the main part of a drawing is modified and therefore gets a new assembly position, the existing drawing is automatically assigned to another part of the position. If the modified part moves to an assembly position that does not have a drawing, the original drawing is automatically cloned to reflect the changes in the modified part.
Holes	The location, size, and number of holes affects numbering.
Part name	The part name affects numbering.
Beam orientation	The orientation of beams affects numbering of assemblies.
Column orientation	The orientation of columns affects numbering of assemblies.
Assembly name	The assembly name affects numbering.
Assembly phase	Only enabled when XS_ENABLE_PHASE_OPTION_IN_NUMBERING (page 245) is set to `TRUE`. The assembly phase affects numbering.
Reinforcing bars	Reinforcing bars affect numbering.
Embedded objects	Sub-assemblies affect the numbering of cast units.
Surface treatment	Surface treatments affect the numbering of assemblies.
Welds	Welds affect the numbering of assemblies.
Tolerance	Parts get the same number if their dimensions differ less than the value entered in this box.
Assembly position sort order	See .
Family numbering	See .

Weld numbering settings

Use the **Weld Numbering** dialog box to view and modify the weld numbering settings. The weld number is displayed in drawings and weld reports.

Option	Description
Start number	The number from which the numbering starts. Tekla Structures automatically suggests the following free number as the start number.
Apply for	Defines which objects are affected by the change. **All welds** changes the number of all welds in the model. **Selected welds** changes the number of the selected welds without affecting others.
Renumber also welds that have a number	Tekla Structures replaces existing weld numbers.
Re-use numbers of deleted welds	If some welds have been removed, Tekla Structures uses their numbers when numbering other welds.

See also

Control number settings

Use the **Create control numbers (S9)** dialog box to view and modify the control number settings.

Option	Description
Numbering	Defines which parts get control numbers. **All** creates consecutive numbers for all parts. **By numbering series** creates control numbers for parts in a specific numbering series.
Assembly/Cast unit numbering series	Defines the prefix and start number of the numbering series for which to create control numbers. Needed only with the **By numbering series** option.

Option	Description
Start number of control numbers	The number from which the numbering starts.
Step value	Defines the interval between two control numbers.
Renumber	Defines how to treat parts that already have control numbers. **Yes** replaces the existing control numbers. **No** keeps the existing control numbers.
First direction	Defines in what order to assign control numbers.
Second direction	
Third direction	
Write UDA to	Defines where to save the control numbers. **Assembly** saves the control numbers to the user-defined attributes of assemblies or cast units. **Main part** saves the control numbers to the user-defined attributes of assembly or cast unit main parts. The control number appears on the **Parameters** tab.

Reinforcement settings

This section provides more information about the various reinforcement settings you can modify in Tekla Structures.

Click the links below to find out more:

Reinforcing bar and bar group properties (page 632)

Reinforcement mesh properties (page 635)

Rebar set properties (page 639)

Reinforcement strand properties (page 655)

Reinforcing bar and bar group properties

Use the **Single rebar** and **Rebar group** properties to view and modify the properties of reinforcing bars and reinforcing bar groups. The file name extension of the property file is:

- `.rbr` for bars
- `.rbg` for groups
- `.rci` for circular groups
- `.rcu` for curved groups

General, Hooks, Cover thickness, IFC export, More

The following properties are available for single reinforcing bars and reinforcing bar groups:

Option	Description	
Name	User-definable name of the bar. Tekla Structures uses bar names in reports and drawing lists, and to identify bars of the same type.	
Grade	Steel grade of the bar.	Size-grade-radius combinations are predefined in the rebar catalog. Click the **...** button to open the **Select rebar** dialog box. The dialog box shows the available bar sizes for the chosen grade. You can also select whether the bar is a main bar, or a stirrup or tie.
Size	Diameter of the bar. Depending on the environment, the nominal diameter of the bar, or a mark that defines the diameter.	
Bending radius	Internal radius of the bends in the bar. You can enter a separate value for each bar bend. Separate the values with spaces. Bending radius complies with the design code you are using. Main bars, stirrups, ties, and hooks usually have their own minimum internal bending radii, which are proportional to the diameter of the reinforcing bar. The actual bending radius is normally chosen to suit the size of the mandrels on the bar-bending machine.	
Class	Use to group reinforcement. For example, you can display bars of different classes in different colors.	

Option	Description	
Numbering	Mark series of the reinforcing bar.	
Hook type	Shape of the hook.	The rebar catalog (`rebar_database.inp`) contains the predefined minimum bending radius and minimum hook length for all standard hooks. See .
Angle	Angle of a custom hook.	
Radius	Internal bending radius of a standard hook or custom hook.	
Length	Length of the straight part of a standard or custom hook.	
Cover thickness on plane	Distances from the part surfaces to the bar on the same plane as the bar.	See .
Cover thickness from plane	Distance from the part surface to the bar, or to the bar end, perpendicular to the bar plane.	
Start	Concrete cover thickness or leg length at the first end of the bar.	
End	Concrete cover thickness or leg length at the second end of the bar.	
IFC entity	For IFC export, select the IFC entity type and subtype of the bar or bar group. The available subtypes depend on the selected IFC entity. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.	
Subtype (IFC4)		
User-defined type (IFC4)		
UDAs	You can create user-defined attributes to add information about reinforcement. Attributes can consist of numbers, text, or lists. You can use the values of user-defined attributes in reports and drawings. You can also change the name of the fields, and add new ones, by editing the `objects.inp` file.	

Special, Distribution

The following properties are available for:

- reinforcing bar groups, including tapered groups
- curved reinforcing bar groups
- circular reinforcing bar groups

Option	Description	
Rebar group type	What is the type of the group.	See .
Number of cross sections		
Creation method	How the bars are spaced.	See .
Number of reinforcing bars		
Target spacing value		
Exact spacing value		
Exact spacing values		
Exclude	Which bars are omitted from the group.	See .

Reinforcement mesh properties

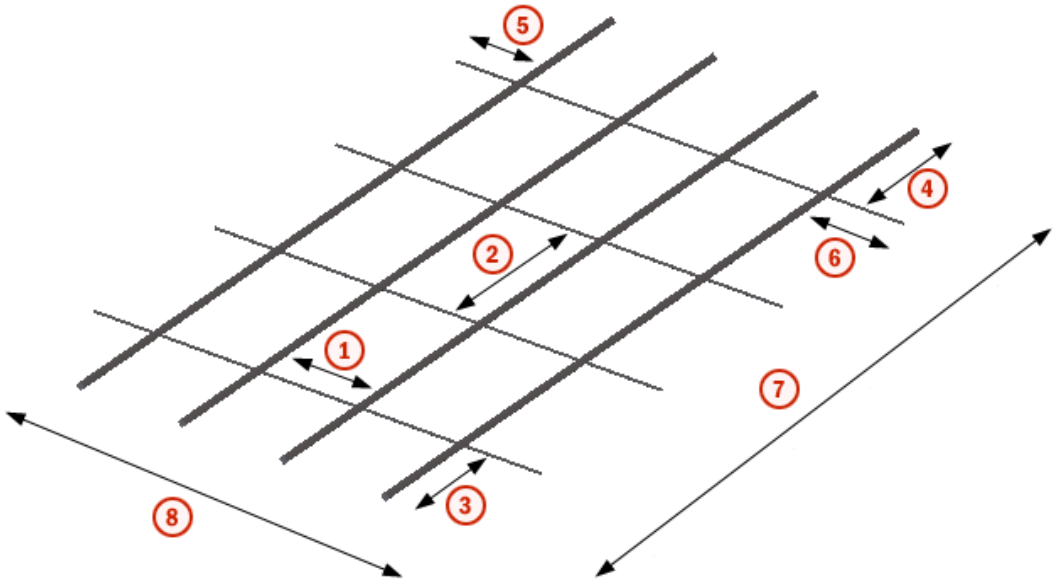
Use the **Rebar mesh** properties to view and modify the properties of reinforcement meshes. The file name extension of a reinforcement mesh property file is `.rbm`.

Option	Description
Name	User-definable name of the mesh. Tekla Structures uses mesh names in reports and drawing lists.
Class	Use to group reinforcement. For example, you can display bars of different classes in different colors.
Numbering	Mark series of the mesh.
Mesh type	Shape of the mesh. Select **Polygon**, **Rectangle**, or **Bent**.
Cross bar location	Define whether the crossing bars are located above or below the longitudinal bars.
Cut by father part cuts	Define whether the polygon or part cuts in the part also cut the mesh.

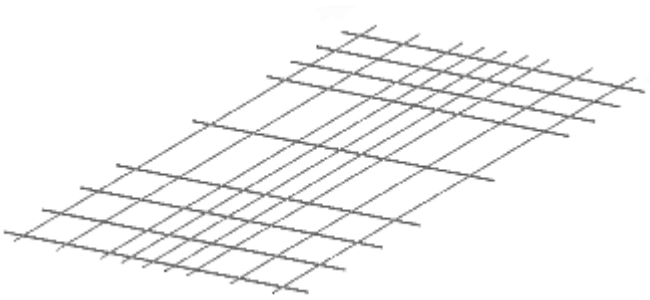
Option	Description
Mesh	Identifier of the mesh. For standard meshes, the mesh name used in the mesh catalog. To create a **Standard** mesh, click the **...** button and select a mesh from the mesh catalog. The properties of standard meshes are defined in the `mesh_database.inp` file. To create a custom mesh, select the **Custom mesh** option and define the properties (page 637).
Grade	Steel grade of the bars in the mesh. Available for custom meshes.
Bending radius	Internal radius of the bends in the bar. Available for bent meshes.
Hooks	See . Available for bent meshes.
Cover thickness on plane	Distance from the part surface to the main bars on the same plane as the bars.
Cover thickness from plane	Distance from the part surface to the bar, or bar end, perpendicular to the bar plane.
Start	Thickness of concrete cover or leg length from the mesh starting point. Available for rectangular and bent meshes.
End	Thickness of concrete cover or leg length at the end point of the bar. Available for bent meshes.
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the mesh. The available subtypes depend on the selected IFC entity. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
UDAs	You can create user-defined attributes to add information about reinforcement. Attributes can consist of numbers, text, or lists. You can use the values of user-defined attributes in reports and drawings. You can also change the name of the fields, and add new ones, by editing the `objects.inp` file.

Custom reinforcement mesh properties
Use the **Rebar mesh** properties to view and modify the properties of customized reinforcement meshes. The file name extension of a reinforcement mesh properties file is `.rbm`.

You can define the following properties for the customized reinforcement meshes:



1. Longitudinal distance
2. Cross distance
3. Longitudinal left overhang
4. Longitudinal right overhang
5. Cross left overhang
6. Cross right overhang
7. Length
8. Width

Option	Description
Spacing method	Define how the mesh bars are distributed. • **Same distance for all**: Use to create meshes with evenly-spaced bars. Tekla Structures distributes as many bars as possible for the length of **Length** or **Width**, using the **Distances** and **Left overhang** values. The **Right overhang** is calculated automatically, and it cannot be zero. • **Multiple varying distances**: Use to create meshes with unevenly-spaced bars. Tekla Structures calculates the **Width** and **Length** based on the **Distances**, the **Left overhang** and the **Right overhang** values. If you do not change any of the values, the spacing method changes back to **Same distance for all**.
Distances	Spacing values of longitudinal or crossing bars. If you select the **Multiple varying distances** spacing method, enter all spacing values, separated by spaces. You can use multiplication to repeat spacing values. For example: `2*150 200 3*400 200 2*150` You can create meshes with unevenly-spaced bars. You can also define a different bar size or multiple different bar sizes for the longitudinal bars and the crossing bars. Multiple bar sizes enable pattern creation. For example, if you enter bar diameters 20 2*6 in the longitudinal direction, Tekla Structures creates a pattern with one size 20 bar and two size 6 bars. This pattern can be repeated in the mesh along the longitudinal direction. 

Option	Description
Left overhang	Extensions of crossing bars over the outermost longitudinal bars. Extensions of longitudinal bars over the outermost crossing bars.
Right overhang	
Diameters	Diameter or size of the longitudinal or crossing bars. You can define multiple diameters for the bars in both directions. Enter all the diameter values, separated by spaces. You can use multiplication to repeat diameter values. For example, `12 2*6` in longitudinal direction and `6 20 2*12` in crossing direction.
Width	Length of crossing bars.
Length	Length of longitudinal bars.
Grade	Steel grade of the bars in the mesh.

See also

Reinforcement mesh properties (page 635)

Rebar set properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar sets. The file name extension of the property file is `.rst`.

General

Option	Description	
Name	User-definable name of the bars. Tekla Structures uses bar names in reports and drawing lists, and to identify bars of the same type.	
Grade	Steel grade of the bars.	Grade-size-radius combinations are predefined in the rebar catalog. Click the **...** button in the property pane to open the **Select rebar** dialog box. The dialog box shows the available bar sizes for the chosen grade. You can also select whether the bars are main bars, or stirrups or ties.
Size	Diameter of the bars. Depending on the environment, the nominal diameter of the bars, or a mark that defines the diameter.	
Bending radius	Internal radius of the bends in the bars. Bending radius complies with the design code you are using. Main bars, stirrups, ties, and hooks usually have	

Option	Description	
	their own minimum internal bending radii, which are proportional to the diameter of the reinforcing bar. The actual bending radius is normally chosen to suit the size of the mandrels on the bar-bending machine. Automatic values are shown in square brackets, for example [120.00].	
Class	Used to group reinforcement. For example, you can display bars of different classes in different colors.	
Numbering	Numbering series of the bars.	

Special

Option	Description
Follow edges	Select whether the rebar set guideline attempts to follow the leg face edges that are located between the guideline end points.
Layer number	Defines the order of the bar layers. The smaller the layer number, the closer to the concrete surface the bar layer is. You can use both positive and negative numbers. If you do not define the layer numbers, Tekla Structures arranges the bar layers according to their creation order. The bar layer that is created first is closest to the concrete surface. You can also adjust the layer order by using these options on the contextual toolbar: Note that if you copy properties from one rebar set to another, the layer number is not copied.

Distribution

Option	Description
Start offset	The offsets at the start and end of the guideline. By default, Tekla Structures calculates the offset values according to the concrete cover settings
End offset	

Option	Description
	and bar diameter. Automatic values are shown in square brackets, for example [32.00]. Note that the automatic offset values may change if the outermost rebar set bars are split by splitters and the split bars would end up in the concrete cover area.
Creation method	How the bars are spaced. For more information, see Spacing options.
Number of reinforcing bars	
Target spacing value	
Exact spacing value	
Exact spacing values	
Exclude	Which bars are omitted from the rebar set. See also Examples.

Advanced: Rounding

Option	Description
Straight bars	Define whether the lengths of straight bars, first and last legs, and intermediate legs are rounded, and whether the bar lengths are rounded up, down, or to the nearest suitable number according to the rounding accuracy.
First and last legs	
Intermediate legs	
Rounding up at splitters	At splitter locations, define how much the bar lengths can be rounded up.

Advanced: Step tapering

Option	Description
Type	Define whether the bars are step tapered, and how the tapering steps are created. The options are **None**, **Distance**, and **Number of bars**.
	If you select the **Number of bars** option, enter the number of bars in one tapering step.
Straight bars	If you select the **Distance** option, enter the tapering step values for straight bars, first and last legs, and intermediate legs.
First and last legs	
Intermediate legs	

Advanced: Minimum lengths to be created

Option	Description
Minimum bar length	Use to prevent Tekla Structures from creating reinforcing bars that are too short. This setting is primarily for straight bars. Enter the minimum bar length as **Distance** or as **Coefficient of bar diameter**.
Minimum straight start/end leg length	Use for bent reinforcing bars. Enter the minimum leg length as **Distance** or as **Coefficient of bar diameter**.

IFC export

For IFC export, select an option in **IFC entity** and in **Subtype (IFC4)**. The available subtypes depend on the selected IFC entity.

You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.

More

Click the **User-defined attributes** button to open the user-defined attributes of the rebar sets. The file name extension of the user-defined attribute file is `.rst.more`.

You can use user-defined attributes to define or override settings, such as bar layer prefix and number, or grouping settings.

See also

Secondary guideline properties (page 642)

Leg face properties (page 643)

Leg surface properties (page 644)

Property modifier properties (page 645)

End detail modifier properties (page 649)

Splitter properties (page 652)

Secondary guideline properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set secondary guidelines.

General

To make a secondary guideline follow the leg face edges that are located between the guideline end points, select **Yes** from the **Follow edges** list in the property pane.

Alternatively, select the secondary guideline and click on the contextual toolbar.

Spacing properties

If you want a secondary guideline to have similar spacing properties as the primary guideline, select **Yes** from the **Inherit from primary** list in the property pane. You can then modify the **Start offset** and **End offset** values if needed. The spacing values and ranges are automatically scaled by the ratio of the secondary guideline length to the primary guideline length.

If you want to define the secondary guideline spacing properties independently from the primary guideline, select **No** from the **Inherit from primary** list, and then modify the following spacing properties as needed:

Option	Description
Start offset	The offsets at the start and end of the guideline. By default, Tekla Structures calculates the offset values according to the concrete cover settings and bar diameter. Automatic values are shown in square brackets, for example [32.00]. Note that the automatic offset values may change if the outermost rebar set bars are split by splitters and the split bars would end up in the concrete cover area.
End offset	
Creation method	How the bars are spaced. For more information, see Spacing options.
Number of reinforcing bars	
Target spacing value	
Exact spacing value	
Exact spacing values	
Exclude	Which bars are omitted from the rebar set. See also Examples.

See also

Rebar set properties (page 639)

Leg face properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set leg faces.

Attributes

Option	Description
Layer number	Defines the order of the bar layers. The smaller the layer number, the closer to the concrete surface the bar layer is. You can use both positive and negative values. By default, Tekla Structures arranges the bar layers according to their creation order. The bar layer that is created first is closest to the concrete surface. You can also adjust the layer order by using these options on the contextual toolbar: Note that if you copy properties from one leg face to another, the layer number is not copied.
Additional offset	Distance between the leg face and the bars. A negative value moves the bars outside the concrete.
Flip bar side	Shows if the bars are flipped over to the other side of the leg face (**Yes**) or not (**No**). The default value is **No**.

See also

Rebar set properties (page 639)

Leg surface properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set leg surfaces. The file name extension of the property file is `.rst_ls`.

Attributes

Option	Description
Layer number	Defines the order of the bar layers. The smaller the layer number, the closer to the concrete surface the bar layer is. You can use both positive and negative values. By default, Tekla Structures arranges the bar layers according to their creation order. The bar layer that is created first is closest to the concrete surface.

Option	Description
	You can also adjust the layer order by using these options on the contextual toolbar: Note that if you copy properties from one leg surface to another, the layer number is not copied.
Additional offset	Distance between the leg surface and the bars. A negative value moves the bars outside the concrete.
Create holes	Select whether the bars at the leg surface are cut by the openings in the concrete.

See also

Rebar set properties (page 639)

Leg face properties (page 643)

Property modifier properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set property modifiers. The file name extension of the property file is `.rst_pm`.

General

Option	Description	
Name	User-definable name of the bars. Tekla Structures uses bar names in reports and drawing lists, and to identify bars of the same type.	
Grade	Steel grade of the bars.	Grade-size-radius combinations are predefined in the rebar catalog. Click the **...** button in the property pane to open the **Select rebar** dialog box. The dialog box shows the available bar sizes for the chosen grade. You can also select whether the bars are main bars, or stirrups or ties.
Size	Diameter of the bars. Depending on the environment, the nominal diameter of the bars, or a mark that defines the diameter.	
Bending radius	Internal radius of the bends in the bars. Bending radius complies with the design code you are using. Main bars, stirrups, ties, and hooks usually have their own minimum internal bending radii, which are	

Option	Description	
	proportional to the diameter of the reinforcing bar. The actual bending radius is normally chosen to suit the size of the mandrels on the bar-bending machine. Automatic values are shown in square brackets, for example [120.00].	
Class	Used to group reinforcement. For example, you can display bars of different classes in different colors.	
Numbering	Numbering series of the bars.	

Special

Option	Description
Follow edges	Select whether the property modifier attempts to follow the leg face edges that are located between the modifier end points.
Modify distribution	Select whether the property modifier affects the bar spacings. If you select **Yes**, the **Distribution** properties become available, and **Bars affected** and **First affected bar** are unavailable.
Bars affected	Select how many bars can be modified in the same location: • **1/1** = all bars are modified in the same cross section. • **1/2** = every second bar is modified in the same cross section. • **1/3** = every third bar is modified in the same cross section. • **1/4** = every fourth bar is modified in the same cross section.
First affected bar	Define which is the first bar to be modified, starting from the first end of the modifier. Enter a positive number, or use the arrow buttons to change the number.
Affect whole bar plane	Select whether the property modifier affects all bars of a rebar set that are in the same plane even

Option	Description
	though some of the bars are not touched by the property modifier (or its projection). For example, to modify bars on opposite sides of an opening using the same modifier, select **Yes**.
Grouping	Select whether and how the bars that are affected by the property modifier are grouped. The options are: • **Automatic**: Bars are grouped according to automatic rules. • **Manual**: Bars are grouped regardless of their geometry or arrangement. Note that this option does not group bars from different sides of splitters. • **No grouping**: Bars are not grouped, but they are individual bars. Use this option for overriding automatic and manual grouping.

Distribution

These properties are available when **Modify distribution** is set to **Yes**.

Option	Description
Start offset	The offsets at the start and end of the modifier. Automatic values are shown in square brackets, for example [32.00]. Note that the automatic offset values may change if the outermost rebar set bars are split by splitters and the split bars would end up in the concrete cover area.
End offset	
Creation method	How the bars are spaced. For more information, see Spacing options.
Number of reinforcing bars	
Target spacing value	
Exact spacing value	
Exact spacing values	
Exclude	Which bars are omitted from the rebar set. See also Examples.

Advanced: Rounding

Option	Description
Straight bars	Define whether the lengths of straight bars, first and last legs, and intermediate legs are rounded, and whether the bar lengths are rounded up, down, or to the nearest suitable number according to the rounding accuracy.
First and last legs	
Intermediate legs	
Rounding up at splitters	At splitter locations, define how much the bar lengths can be rounded up.

Advanced: Step tapering

Option	Description
Type	Define whether the bars are step tapered, and how the tapering steps are created. The options are **None**, **Distance**, and **Number of bars**.
	If you select the **Number of bars** option, enter the number of bars in one tapering step.
Straight bars	If you select the **Distance** option, enter the tapering step values for straight bars, first and last legs, and intermediate legs.
First and last legs	
Intermediate legs	

Advanced: Minimum lengths to be created

Option	Description
Minimum bar length	Use to prevent Tekla Structures from creating reinforcing bars that are too short. This setting is primarily for straight bars. Enter the minimum bar length as **Distance** or as **Coefficient of bar diameter**.
Minimum straight start/end leg length	Use for bent reinforcing bars. Enter the minimum leg length as **Distance** or as **Coefficient of bar diameter**.

More

Click the **User-defined attributes** button to open the user-defined attributes of the rebar set property modifiers. The file name extension of the user-defined attribute file is `.rst_pm.more`.

You can use user-defined attributes to define or override settings, such as bar layer prefix and number, or grouping settings.

See also

Rebar set properties (page 639)

End detail modifier properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set end detail modifiers. The file name extension of the property file is `.rst_edm`.

Special

Option	Description
Follow edges	Select whether the end detail modifier attempts to follow the leg face edges that are located between the modifier end points.
Bars affected	Select how many bars can be modified in the same location: • **1/1** = all bars are modified in the same cross section. • **1/2** = every second bar is modified in the same cross section. • **1/3** = every third bar is modified in the same cross section. • **1/4** = every fourth bar is modified in the same cross section.
First affected bar	Define which is the first bar to be modified, starting from the first end of the modifier. Enter a positive number, or use the arrow buttons to change the number.
End type	Select **Hook** or **Cranking**. If you select the empty option, no hooks or cranks are created, but you can define length adjustments, end preparations, and user-defined attributes.

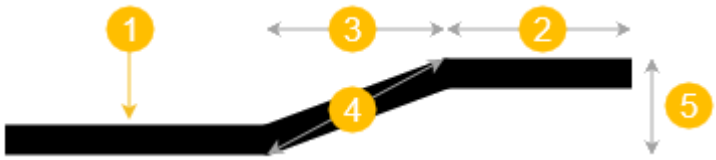
Hook

These properties are available when **End type** is **Hook**.

Option	Description	
Hook type	Shape of the hook.	The rebar catalog (`rebar_database.inp`) contains the predefined minimum bending radius and minimum hook length for all standard hooks. See .
Angle	Angle of a custom hook.	
Radius	Internal bending radius of a standard hook or custom hook.	
Length	Length of the straight part of a standard or custom hook.	
Hook rotation	Rotation angle of a hook out of the bar plane. Use to create 3D bars.	For example:

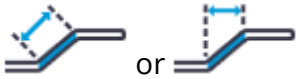
Cranking

These properties are available when **End type** is **Cranking**.



(1) = Location of the end detail modifier

Option	Description
Cranking type	Select **No cranking**, **Standard cranking**, or **Custom cranking**. Use the **No cranking** option to override other end detail modifiers that create cranks. With standard cranking, the crank dimensions are read from the rebar catalog (`rebar_database.inp`).
Crank straight length	With custom cranking, enter the length of the straight segment of the crank. This is **(2)** in the image above.
Cranked length	With custom cranking, select whether the length of the cranked segment is defined in the diagonal **(4)** or horizontal **(3)** direction:

Option	Description
	 or Then select and enter the needed distance or a multiplier of the bar diameter.
Cranked offset	With custom cranking, enter the offset distance of the straight segment of the crank. This is **(5)** in the image above. The default value is 2 * actual bar diameter.
Crank rotation	Defines to which angle the crank is rotated.

Length adjustment

Option	Description
Adjustment type	Select whether and how the bar length is adjusted (extended or shortened). • **No adjustment**: Bar length is not adjusted. • **End offset**: Bar length is adjusted according to the specified end offset. Use this option to keep the leg faces on the concrete faces and adaptive to the concrete faces, but still to extend or shorten the bar ends. • **Leg length**: Bar length is adjusted according to the specified leg length.
Length	Depending on the adjustment type, the length of the end offset or leg. With end offset, enter a positive value to extend the bars, or a negative value to shorten the bars. With leg length, enter a positive value to set the leg length.
Align bar ends	When the lengths of straight bars are rounded and/or step tapered, select whether the bar ends nearest to the end detail modifier are aligned or not. If you select **No**, the rounding and step tapering happens at the tapered edge of the rebar set, and if both edges are tapered, at the edge that has a larger angle.

End preparations

Option	Description
Method	Select the end method of the bars. The options are: • **Coupler** • **Coupler female** • **Coupler male** • **Threaded** • **Anchor**
Type	Select the end method type. The options are: • **Standard** • **Position** • **Bridging** • **Transition** • **Bolt** • **Weldable**
Product	Product name of the end detail. Can be shown in reports.
Code	Product code of the end detail. Can be shown in reports.
Thread type	Enter the type of the threading.
Threaded length	Length of the threading from the bar end.
Extra fabrication length	Additional length needed with some threading methods. Can be shown in reports, but does not affect the total length of the bar.

More

Click the **User-defined attributes** button to open the user-defined attributes of the rebar set end detail modifiers. The file name extension of the user-defined attribute file is `.rst_edm.more`.

See also

Rebar set properties (page 639)

Splitter properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set splitters. The file name extension of the property file is `.rst_sm`.

Some of the following settings depend on the splitter direction. An arrowhead symbol close to the midpoint of each splitter indicates the direction and the left and right sides of the splitter. The arrow points from the start towards the end of the splitter.

Special

Option	Description
Follow edges	Select whether the splitter attempts to follow the leg face edges that are located between the splitter end points.
Bars affected	Select how many bars can be modified in the same location: • **1/1** = all bars are modified in the same cross section. • **1/2** = every second bar is modified in the same cross section. • **1/3** = every third bar is modified in the same cross section. • **1/4** = every fourth bar is modified in the same cross section.
First affected bar	Define which is the first bar to be modified, starting from the first end of the modifier. Enter a positive number, or use the arrow buttons to change the number.
Split type	Select **Lapping** or **Cranking**.
Split offset	Defines how far from the splitter the split is created. Positive values move the split to the right side and negative values to the left side of the splitter.

Lapping

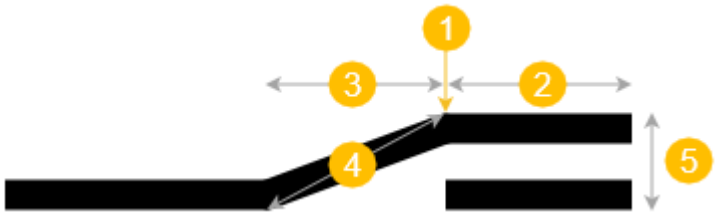
These properties are available when **Split type** is **Lapping**.

Option	Description
Lapping type	Select **Standard lapping** or **Custom lapping**.
Lap length	With custom lapping, enter the length of the lap splice. With standard lapping, the lap length is read from the rebar catalog (`rebar_database.inp`).
Lap side	Select the side of the lap splice from the splitter: • **Lap left**

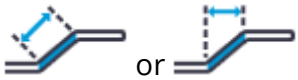
Option	Description
	• **Lap right** • **Lap middle**
Lap placement	Select whether the lapping bars are parallel to each other, or on top of each other.

Cranking

These properties are available when **Split type** is **Cranking**.



(1) = Location of the splitter

Option	Description
Cranking type	Select **Standard cranking** or **Custom cranking**. With standard cranking, the crank dimensions are read from the rebar catalog (`rebar_database.inp`).
Crank straight length	With custom cranking, enter the length of the straight segment of the crank. This is **(2)** in the image above.
Cranked length	With custom cranking, select whether the length of the cranked segment is defined in the diagonal **(4)** or horizontal **(3)** direction:  or Then select and enter the needed distance or a multiplier of the bar diameter.
Cranked offset	With custom cranking, enter the offset distance of the straight segment of the crank. This is **(5)** in the image above. The default value is 2 * actual bar diameter.
Crank side	Select to which side of the splitter the crank is created, **Left** or **Right**.
Crank rotation	Defines to which angle the crank is rotated.

Staggering

Option	Description
Stagger type	Select whether and in which direction the splices are staggered. The options are: • **No stagger** • **Stagger left** • **Stagger right** • **Stagger middle**
Stagger offset	The offset of the adjacent bars if they are staggered.

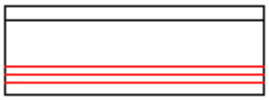
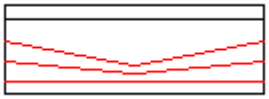
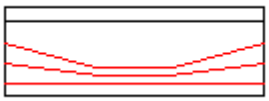
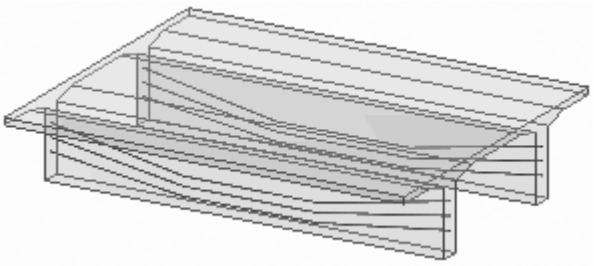
See also

Rebar set properties (page 639)

Reinforcement strand properties

Use the **Strand pattern** properties to view and modify the properties of strands. The file name extension of the property file is `.rbs`.

Option	Description
General	
Name	User-definable name of the strand. Tekla Structures uses strand names in reports and drawing lists, and to identify strands of the same type.
Grade	Steel grade of the strand.
Size	Diameter of the strand. Depending on the environment, the nominal diameter of the strand, or a mark that defines the diameter.
Bending radius	Internal radius of the bends in the strand. You can enter a separate value for each bend. Separate the values with spaces.
Class	Use to group reinforcement. For example, you can display strands of different classes in different colors.
Numbering	Mark series of the strand.
Special	
Pull per strand	Pre-stress load per strand (kN).

Option	Description
Number of cross sections	Number of cross sections of the strand pattern. For example: • Number of cross sections along strand profile = 1:  • Number of cross sections along strand profile = 2: • Number of cross sections along strand profile = 3:  • Number of cross sections along strand profile = 4:  In this double-tee beam, the number of cross sections is 4: 
Debonding	
Debonded strands	Enter the strand number. The strand number is the selection order number of the strand.

Option	Description
From start **Middle to start** **Middle to end** **From end**	Enter the length of the debonding. If you select the **Symmetry** check box, values from **From start** and **Middle to start** are copied to **From end** and **Middle to end**.
Symmetry	Define whether the end and start lengths are symmetrical.
IFC export	
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the strands. The available subtypes depend on the selected IFC entity. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
More	
UDAs	You can create user-defined attributes to add information about reinforcement. Attributes can consist of numbers, text, or lists. You can use the values of user-defined attributes in reports and drawings. To set values for user-defined attributes, click the **User-defined attributes** button. You can also change the name of the fields, and add new ones, by editing the `objects.inp` file.

3.2 Licensing tools settings

This section contains detailed reference information about the user interface of the following licensing tools:

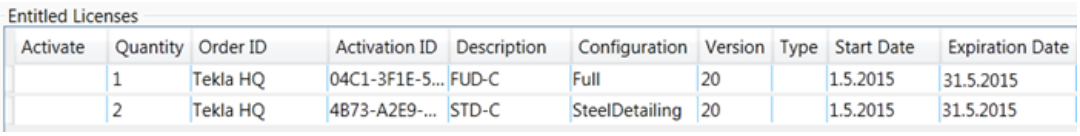
- Tekla License Administration Tool options and settings (page 657)
- Tekla License Borrow Tool options and settings (page 660)
- LMTOOLS options and settings used in Tekla licensing (page 661)

Tekla License Administration Tool options and settings

Tekla License Administration Tool displays information about entitled licenses, activated licenses and license use. You can also activate, deactivate, and repair licenses in the tool.

Entitled licenses

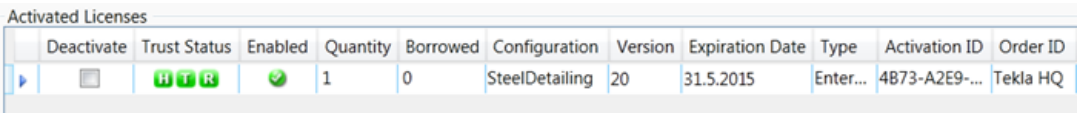
The **Entitled licenses** area on the **Licenses** tab displays information on the licenses your are entitled to. Here you can also activate licenses.



Column	Description
Activate	Used for activating licenses. Displays the number of licenses to be activated.
Quantity	The number of total licenses of the configuration.
Order ID	The order ID of the license. The order ID helps to identify which licenses are linked to each other and which license needs to be deactivated before a new license can be activated.
Activation ID	The activation ID of the license.
Description	The abbreviation of the configuration.
Configuration	The name of the configuration.
Version	The version number of the configuration.
Type	Indicates whether the license is for domestic or enterprise use.
Start Date	The date when the license came into effect.
Expiration Date	The date when the license expires. If the license is permanent, the word `Permanent` is shown instead of the expiration date.

Activated Licenses

The **Activated Licenses** area on the **Licenses** tab displays information about activated licenses. Here you can also deactivate licenses.

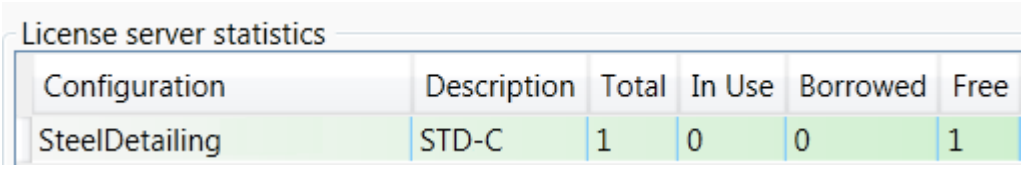


Column	Description
▶	Click the ▶ button to view the following information:

Column	Description
	• **Fulfillment ID**: A unique transaction id which is used in mapping license activations and deactivations together. • **Borrow Info**: On which computer the license is borrowed and when the borrowing period ends. • **Start Date**: When the license starts to work. • **Borrowed**: Number of borrowed licenses.
Deactivate	Deactivate licenses.
Trust Status	The Host (**H**), Time (**T**) and Restore (**R**) Trust Status. Green symbol indicates that the information is trusted. Red symbol indicates that the information is not trusted. If any of these are not trusted, the license cannot be used. **Host** indicates whether the server has been moved from one computer to another or whether the computer hardware has radically changed. **Time** indicates whether the system clock has been tampered with. **Restore** indicates whether the license is obtained from a backup copy.
Enabled	Indicates whether the license is enabled or disabled. For example, a license may be disabled if there is a disconnection during the activation or deactivation. A disabled license cannot be used.
Quantity	The number of activated licenses of the configuration.
Borrowed	The number of borrowed licenses.
Configuration	The name of the configuration.
Version	The version number of the configuration.
Expiration Date	The date when the license expires. If the license is permanent, the word `Permanent` is shown instead of the expiration date.
Type	Indicates whether the license is for domestic or enterprise use.
Activation ID	The activation ID of the license.
Order ID	The order ID of the license. The order ID helps to identify which licenses are linked to each other and which license needs to be deactivated before a new license can be activated.

License server statistics

The **License server statistics** area on the **Statistics** tab displays information about license use.

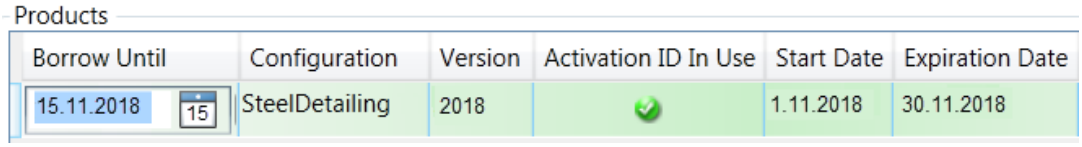


Column	Description
Configuration	The name of the configuration.
Description	The abbreviation of the configuration.
Total	The number of activated licenses.
In Use	The number of licenses in use.
Borrowed	The number of borrowed licenses.
Free	The number of free licenses.

Tekla License Borrow Tool options and settings

Tekla License Borrow Tool displays information about licenses that are available for borrowing and the licenses that have been borrowed. You can also borrow and return licenses

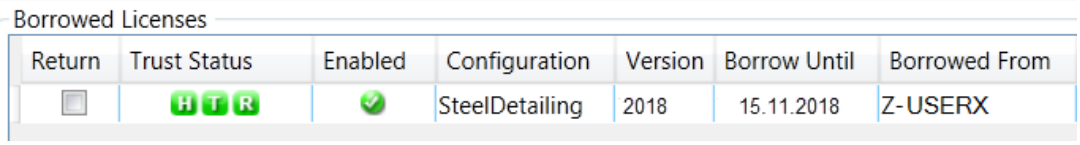
The **Products** area displays information on licenses, and allows you to set the expiration date for the license borrowing:



Option/ Setting	Description
Borrow Until	Select an expiration date for a borrowed license. One month is the maximum.
Configuration	The name of the configuration that you are borrowing.
Version	The version number of the configuration.
Activation ID In Use	Indicates whether the activation ID is used for borrowing. The box is selected only if you are using a customized and exported product ID file.
Start Date	The date when the license came into effect.

Option/ Setting	Description
Expiration Date	The date when the license expires.

The **Borrowed Licenses** area displays information on borrowed licenses, and allows you to return borrowed licenses.



Option/ Setting	Description
Return	Used for returning licenses.
Trust Status	The Host (**H**), Time (**T**) and Restore (**R**) Trust Status. Green symbol indicates that information is trusted. Red symbol indicates that information is not trusted. If any of these are not trusted, the license cannot be used. • **Host** indicates whether the server has been moved from one computer to another or the computer hardware has radically changed. • **Time** indicates whether the system clock has been tampered with. • **Restore** indicates whether the license is obtained from a backup copy.
Enabled	Indicates whether the license is enabled or disabled. For example, a license may be disabled if there is a disconnection during the borrowing. A disabled license cannot be used.
Configuration	The name of the configuration.
Version	The version number of the configuration.
Borrow Until	The date when the borrowing expires.
Borrowed From	The license server from where the license is borrowed.

LMTOOLS options and settings used in Tekla licensing

LMTOOLS is a graphical user interface that allows you to administer the license server. This utility is provided by Flexera Software.

NOTE If you are using **FlexNet Manager** by Flexera Software for managing licenses, do not use **LMTOOLS**.

In addition to Tekla licenses, you can also manage licenses of other software that are using FlexNet or FLEXlm. You can also run **LMTOOLS** on client computers to check the status of the licenses on the license server.

Go to **Tekla Licensing** --> **LMTOOLS** through the **Start** menu or **Start screen**, depending on your Windows operating system.

Service/License File tab

Option/Setting	Description
Configuration using License File	Not used.
Configuration using Services	You must always have **Configuration using Services** and Tekla Licensing Service selected on the **Service/License File** tab when you manage Tekla licenses. These options are selected by default during the installation of the Tekla license server. If you need to configure the license server manually, see .
LMTOOLS ignores license path environment variables	Not used.

System Settings tab

The **System Settings** tab displays common device and operating system information about the computer on which you are running **LMTOOLS**. All the relevant information on licensing is available. For example, you can check the hostname of your server computer.

Option/Setting	Description
Hostid Settings	**Computer/Hostname** **Username** **CPU ID** **IP Address** **Etnernet Address** **Disk Volume Serial Number** **FLEXID**
Time Settings	**System Time Zone** **GMT Time** **Difference from UTC**

Option/Setting	Description
	MSDOS Time **Local Time** **Windows Directory**
Save HOSTID info to a file	Save the hostid information to a text file.

Utilities tab

You can affect the information that is displayed in the status list on the **Server Status** tab by defining values on the **Utilities** tab. By default, the status list displays information about all the license servers you are connected to.

Option/Setting	Description
File Name **Browse** **Find Version**	Find out the version of FlexNet Licensing linked with a file. Useful for diagnosing errors. Browse for the file and click **Find Version**.
Vendor Name **Path** **Add Vendor Path** **Override Path** **List All Vendor Paths**	An example of the vendor name and path: **Vendor Name**: `tekla` **Path**: `27007@myserver` (port and license server computer name/hostname) The **Vendor Name** that is used for Tekla license server is `tekla` (all letters lowercase). Add a licensing service to be listed in the status list on the **Server Status** tab, enter the vendor information in the **Vendor Name** and **Path** boxes and click **Add Vendor Path**. If you want the status list to only display information on certain license servers, enter the vendor information in the **Vendor Name** and **Path** boxes and then click **Override Path** to replace the existing license servers displayed in the status list. The **List All Vendor Paths** button lists all the licensing services that are displayed in more detail in the status list on the **Server Status** tab.

Start/Stop/Reread tab

On the **Start/Stop/Reread** tab, you can stop and start the license server, and adjust some setting related to stopping the server.

NOTE When you perform actions concerning the Tekla license server, Tekla Licensing Service must be active in the **FlexNet license services**

installed on this computer list. You can select Tekla Licensing Service from the services list on the **Service/License File** tab.

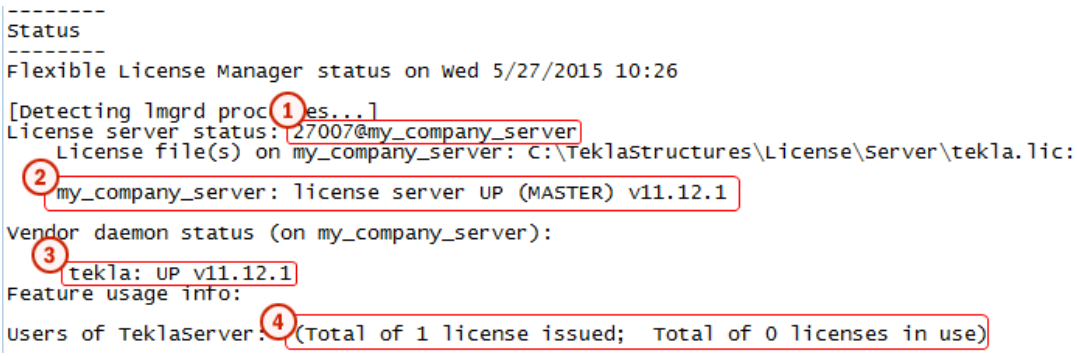
Option/Setting	Description
FlexNet license services installed on this computer	Lists all FlexNet license services available on the license server computer. Always ensure that Tekla Licensing Service is active in the **FlexNet license services installed on this computer** list when you perform actions concerning the Tekla license server. • Activate Tekla Licensing Service by selecting it from the services list on the **Service/License File** tab.
Start Server	Starts the license server that is active in the **FlexNet license services installed on this computer** list.
Stop Server	Stops the license server that is active in the **FlexNet license services installed on this computer** list. When you click the button, the status bar displays the message **Stopping the Server**, and it takes a couple of seconds to stop the license server. The message does not change when the server is stopped. If **Stop Server** does not stop the server, select the **Force Server Shutdown** check box and click **Stop Server** again.
ReRead License File	Updates the license server without stopping and starting it. You need to use this button, when you manually notify the license server about license changes. For more information, see .
Advanced settings	
Restrict lmdown to work only from node where lmgrd is running.	When this option is selected, you can stop the server only on the server computer. No-one can accidentally stop the license server on a client computer. We recommend that you use this option.
Disable lmdown utility, use task manager.	When this option is selected, you cannot stop the server in **LMTOOLS**. The server can only be stopped in Windows Task Manager.
Disable 'lmremove' of license file.	Not in use in Tekla licensing.

Server Status tab

The **Server Status** tab displays the status of the license server and the licenses. Here you can check how many licenses are in use or borrowed, who is currently using licenses on the server and on which computers licenses have been borrowed.

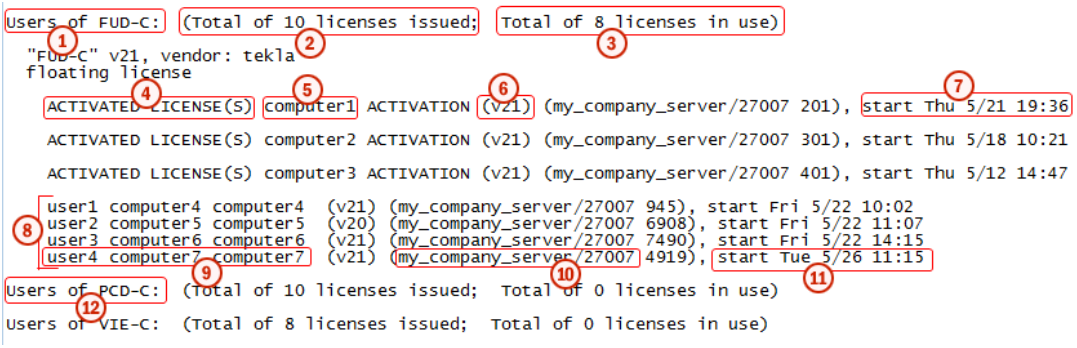
If you want to check the status of the Tekla license server and licenses only, enter `tekla` in the **Individual Daemon** box and click **Perform Status Enquiry**.

Below is an example of license server information:



1. The license server port@hostname
2. The license server is up and running
3. License server with Tekla identity is up and running
4. A default value for TeklaServer displayed in every status check

Below is an example of license information:



1. Indications the configuration for which license information is displayed. In this case, Full Detailing.
2. The number of activated licenses on the license server
3. The number of the licenses in use; checked out from the license server or borrowed
4. A borrowed license
5. On which computer the license was borrowed

6. Tekla license version
7. Time when the license was borrowed
8. Licenses checked out from the license server
9. On which computer and display the user has checked the license out. In this case, the user is user4, the computer and display names are computer7.
10. The license server hostname/port from where the license is checked out
11. Time when the Tekla Structures session was started
12. Usage of licenses of other configurations. In this case, Precast Concrete Detailing.

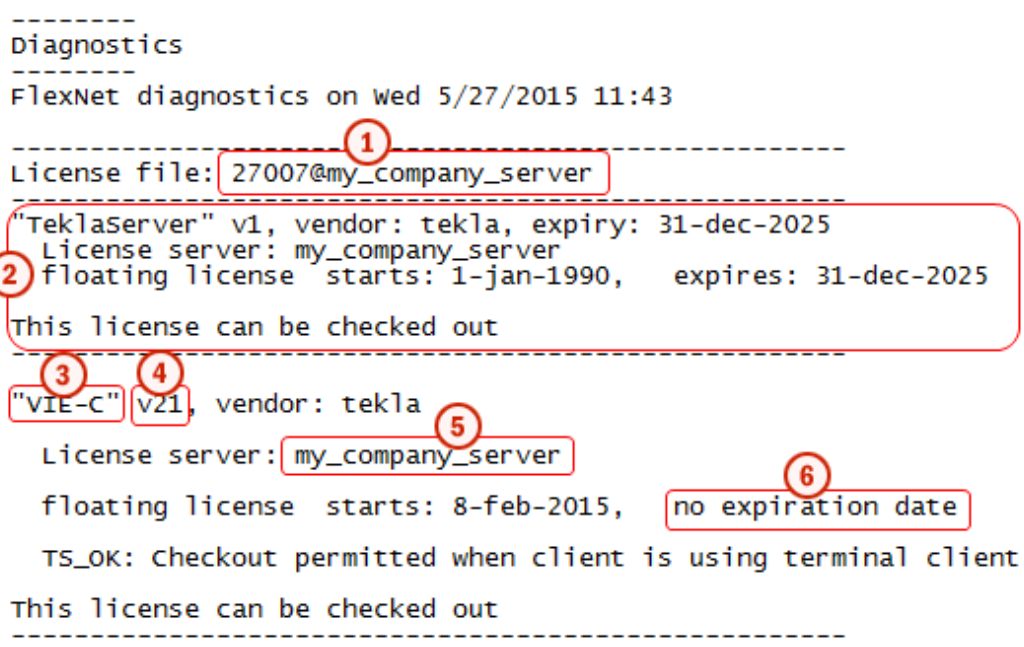
Server Diags tab

The **Server Diags** tab displays more information about the license servers and licenses, and provides diagnostics. For more information about the licenses and their status, see Tekla License Administration Tool options and settings (page 657).

NOTE If you have problems with the license server, send a copy of the `tekla.lic`, `tekla.opt`, and `tekla_debug.log` files to your local Tekla Structures support. The information on the **Server Diags** tab is not detailed enough to solve some of the problems.

- To display the diagnostics, click **Perform Diagnostics**.

Below is an example of the **LMTOOLS** license server and license diagnostics:



1. The license server port@hostname

2. Common information for all Tekla license server installations
3. On which configuration the information is displayed. In this case it is Viewer.
4. Tekla license version
5. The license server hostname
6. The expiration date of the license. In this case, the license is permanent.

Config services tab

The values on the **Config Services** tab are filled in automatically during the installation of the Tekla license server. However, if you encounter problems during the installation and the license server does not start automatically, you need to configure the settings manually on the **Config Services** tab.

For more information about the data you should add/select on this tab, see .

NOTE Whenever you make modifications or perform any actions in **LMTOOLS** concerning the Tekla license server, you need to have Tekla Licensing Service selected in the **Service Name** list.

Borrowing tab

WARNING Use Tekla License Borrow Tool for borrowing Tekla licenses. Do not use the Borrowing tab of **LMTOOLS** with Tekla licenses.

3.3 Drawing settings reference

There are a lot of settings in Tekla Structures that you mainly control in the properties dialog boxes. There are also some additional settings files that you need to modify in a text editor.

Click the links below to find out more:

- General arrangement drawing properties (page 668)
- Single-part, assembly and cast unit drawing properties (page 671)
- Layout properties (page 674)
- View properties in drawings (page 676)
- Section view properties (page 682)
- Dimension and dimensioning properties in drawings (page 684)
- Mark properties in drawings (page 733)
- Mark elements in drawing marks (page 740)
- Pour object and pour break properties in drawings (page 781)

- Part and neighbor part properties in drawings (page 759)
- Bolt properties in drawings (page 764)
- Surface treatment properties in drawings (page 765)
- Surface treatment hatch pattern properties (surfacing.htc) (page 766)
- Reinforcement object properties in drawings (page 768)
- Reinforcement settings for drawings (rebar_config.inp) (page 775)
- Placement properties for marks, dimensions, notes, texts and symbols in drawings (page 784)
- Model weld properties in drawings (page 786)
- Drawing sketch object properties (page 787)
- Drawing grid and grid line properties (page 789)

General arrangement drawing properties

On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** . You can modify the properties after creating the drawing by double-clicking the drawing background.

The options in the general arrangement drawing properties dialog box are described below.

Option	Description	For more information
Name	The name of the drawing. This name is shown in **Document manager**, and can be included in drawing and report templates. The maximum length of the drawing name is 32 characters.	
Title 1, **Title 2**, **Title 3**	Titles are shown in the **Document manager** and in drawing and report templates.	
Use detailed object level settings	Set to **Yes** to use the detailed object level settings created in the **Object level settings for drawing** dialog box.	
Layout...	Select the drawing layout and define drawing sizes. You can	Define drawing size and drawing view scale

Option	Description	For more information
	also select whether you want to list hidden objects in templates.	
View...	Define view properties: scale, neighbor part view extension, reflected view, opening and recess symbol, datum point elevation, part shortening, view label, and anchor bolt plan settings.	View properties in drawings (page 676)
Detail view...	Define detail view properties: view label, detail boundary and detail mark settings.	
Dimension...	Define dimension properties: dimension type, units, precision, format, placing, and appearance.	Dimension and dimensioning properties in drawings (page 684)
Dimensioning...	Define dimensioning properties: grid dimension and part dimension settings.	Dimension and dimensioning properties in drawings (page 684) General arrangement drawing dimensioning properties (page 729)
Part mark... **Bolt mark...** **Neighbor part mark...** **Surface treatment mark...** **Weld mark...** **Reinforcement marks...** **Neighbor reinforcement marks...** **Connection mark...** **Pour object mark...**	Define mark properties: included elements and element settings, and mark visibility, mark frame, mark leader line and placing settings.	Mark properties in drawings (page 733) Mark properties - Content, General, Merging and Appearance tabs (page 734) Drawing weld mark properties (page 750) Model weld mark properties in drawings (page 753) Mark elements (page 740)

Option	Description	For more information
Part...	Define part properties: part representation, hidden line, center line and reference line visibility, additional mark visibility, part appearance, and fill settings.	Part and neighbor part properties in drawings (page 759)
Bolt...	Define bolt properties: bolt representation, bolt symbol content, bolt visibility, and bolt appearance settings.	Bolt properties in drawings (page 764)
Neighbor part...	Define neighbor part properties: visibility, part representation, hidden line, center line and reference line visibility, additional mark visibility, and part appearance settings. You can also define bolt representation and bolt symbol content settings for neighbor parts.	Part and neighbor part properties in drawings (page 759)
Surface treatment...	Define surface treatment properties: surface treatment visibility, pattern visibility, hidden lines visibility and surface treatment representation settings.	Surface treatment properties in drawings (page 765)
Weld...	Define welding properties: weld visibility, weld size limit and weld appearance settings.	Model weld properties in drawings (page 786)
Reinforcement...	Define reinforcement and mesh properties: bar and mesh representation and visibility, bar and mesh symbol, and bar and mesh appearance settings.	Reinforcement object properties in drawings (page 768)
Neighbor reinforcement...	Define neighbor reinforcement and mesh properties: bar and mesh representation and visibility, bar and mesh symbol, and bar and mesh appearance settings.	Reinforcement object properties in drawings (page 768)

Option	Description	For more information
Reference objects...	Define reference object visibility and appearance settings.	
Grid...	Define grid visibility and appearance settings.	Drawing grid and grid line properties (page 789)
Protection...	Define protected areas to prevent text, marks or dimensions from being placed there.	
Filter... and **Neighbor part filter...**	Create and modify part filters/ neighbor part filters on drawing level.	
User-defined attributes...	Add customized information to a drawing, such as information related to workflow, and comments. This information can be displayed in **Document manager**, and you can use it in report and drawing templates and marks, and as switches when customizing print file names. The user-defined attributes and tabs shown in this dialog box are defined in the `objects.inp` file. The availability of the various user-defined attributes depends on your environment, role and configuration.	

Single-part, assembly and cast unit drawing properties

On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type. You can modify the properties after creating the drawing by double-clicking the drawing background.

The options in single-part, assembly and cast unit drawing properties dialog box are described below.

Option	Description	For more information
Name	Define the name of the drawing that is shown in the **Document manager**, and that can be included in drawing and report templates. The maximum length of the drawing name is 32 characters.	Modify drawing names and titles
Cast unit definition method	This setting is available for cast unit drawings only. **By cast unit position**: A drawing will be created from each cast unit. If there are identical cast units, one of them will serve as the base cast unit for the drawing. This is the most common method for creating cast unit drawings. **By cast unit ID**: Each part in the model has a unique GUID. You can create drawings by using the cast unit GUIDs. The GUID determines the marking of the drawing. You can create several drawings from identical cast units.	
Title 1, **Title 2**, **Title 3**	Define titles that are shown in **Document manager** and that can be included in drawing and report templates.	
Sheet number	Used for creating several drawings of the same part as multiple drawing sheets. The number of sheets is not limited.	
Layout	Select the drawing layout, and define drawing sizes, autoscale, projection type, view alignment and part expansion settings. You can	Define drawing size and drawing view scale

Option	Description	For more information
	also select whether you want to list hidden objects in templates.	

View creation	Define the drawing views to be created. From here you can move on to modifying the view properties for each view.
	
	Here you can also define **Common settings for all views**:
	
	• **Weld mark visible** controls the visibility of model weld marks in assembly drawings. The options are:
	  • **In one view**: Tekla Structures automatically finds the most relevant view for showing the model weld marks. Every weld mark is shown in a drawing in one view only.
	  • **In all views**: Tekla Structures adds the model weld marks in all drawing views that contain the part with the weld.
	• **Ignore bolt size** filters standard-sized bolts out of drawings, which means that Tekla Structures does not display bolt marks of the defined bolt size in drawings. This setting is available in single-part and assembly drawings.
	• **Weld size limit** filters welds and weld marks of the defined weld size and smaller out of the drawing. This setting is available in
		View properties in drawings (page 676)
		
		Define the views to create

Option	Description	For more information
	single-part and assembly drawings.	
Section view	Define section view properties.	Section view properties (page 682)
Detail view	Define the start number or letter of the detail view and detail symbol label.	
User-defined attributes	Add customized information to a drawing, such as information related to workflow, and comments. This information can be displayed in the **Document manager**, and you can use it in report and drawing templates and marks, and as switches when customizing print file names. The user-defined attributes and tabs shown in this dialog box are defined in the `objects.inp` file. The availability of the various user-defined attributes depends on your environment, role and configuration.	

Layout properties

On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type. Then click **Layout**. You can modify the properties after creating the drawing by double-clicking the drawing background.

All options in all layout property panels and dialog boxes are described below. All the described options are not available for all drawing types.

Option	Description
Drawing size tab	
Layout	Define the layout that you want to use.
Edit	Open the **Layout editor** to customize drawing layouts.

Option	Description
List hidden objects in templates	Select **Yes** to list hidden objects in templates. **No** removes all information about the hidden parts, also from the total weight.
Size definition mode	Select **Autosize** if you want to let Tekla Structures find appropriate sizes and table sets for drawings. Select **Specified size** to specify the exact size for the drawing. The drawing size should always be smaller than the actual paper size to accommodate printer margins.
Drawing size	If you selected **Specified size**, define the drawing size here.
Scale tab	
Autoscale	Set **Autoscale** to **Yes** to let Tekla Structures automatically select the suitable scale for the drawing view.
Main view scales **Section view scales**	When you use autoscaling, enter the denominators of main view and section view scales and separate them with spaces. For example, enter "5 10 15 20" for the scales 1/5, 1/10, 1/15, and 1/20
Scale change mode	When you use autoscaling, set the scale change mode which defines the relationship between the scales of main and section views within a drawing: **main = section**: The scales of the main and section view are equal. **main < section**: Main view scales are smaller than section view scales. **main <= section**: Main view scales are smaller than or equal to section view scales
Preferred size	Enter the preferred size of the drawing, if you use both autosizing and autoscaling. Tekla Structures tries to find a drawing size where the drawing contents fit by first trying to use the exact scale and the smallest drawing size. If the contents do not fit, Tekla Structures increases the drawing size until it reaches the preferred size.
Other tab	
Projection type	Define how Tekla Structures places the projections of a part in cast unit, single-part, and assembly

Option	Description
	drawings. Projection type affects the order of the views in the drawing. The options are: **First angle**, also referred to as the European projection. **Third angle**, also referred to as the American projection.
Align section views with main view **Align end views with main view**	Set to **Yes** to place the views next to the main view. If you select **No**, Tekla Structures places the section and end views in any available location.
Expand shortened parts to fit	Set to **Yes** to stretch shortened views to fill up empty areas of the drawing.
Include single-parts	Set to **Yes** to include in assembly drawings the single-part drawings of the individual parts that make the assembly. Setting this to **Yes**, activates the **Single-part attributes** option.
Single-part attributes	Define the desired single-part drawing properties to be used in the single-part view. To do this, set **Include single-parts** to **Yes**.

View properties in drawings

Use the **View Properties** dialog box to view and modify the drawing view properties.

The table below describes all view level properties for all drawing types.

Option	Description
Views tab when **View creation** is selected from the options tree in drawing properties:	
View type on/off	Defines the main views, section views and 3D views that you want to create. • If you select **Off**, Tekla Structures does not create the view, but dimensions the parts in the available views. If you set all four main views off, Tekla Structures will still create a front view. • If you select **On**, Tekla Structures always creates the view, even if it was not necessary in order to

Option	Description
	show the dimensions. For section views, Tekla Structures creates one additional section view showing the middle of the main part. For end views, Tekla Structures creates end view from one end of the main part. • If you select **Auto**, Tekla Structures creates the view if it is necessary in order to show the dimensions. For section views, Tekla Structures creates the necessary number of views to show all the dimensions. For end views, Tekla Structures also creates another end view from the other end of the main part, if there are dimensions at that end.
View label	Shows the view label set in view properties. If the label is defined in view properties, you can change it here. You can always change the label of the main views.
View properties	Shows the current view properties for the selected view. You can select another view properties file from the list and modify the view properties by clicking **View properties**. Note that if you have selected **Auto** in the **on/off** column as the creation method, Tekla Structures will use the standard properties even if you define another property file here.
Attributes tab when **View creation** is selected from the options tree in drawing properties:	
Coordinate system	Sets the coordinate system of the drawing views. The options are: local, model, oriented, horizontal brace, vertical brace, and fixed. For more information, see Change the coordinate system.

Option	Description
Rotate coordinate systemAround X,**Around Y** and **Around Z**	Rotates the view around the x, y, or z axis of the parts using the entered values. For more information, see Rotate parts in drawing views.
Unfolded	**Yes** shows and dimensions the bend lines in the drawing. For more information, see Unfold polybeams in drawings.
Undeformed	**Yes** undeforms deformed parts and shows the developed (undeformed) shape of the deformed parts in drawings. For more information, see Undeform deformed parts in drawings.
Recreate the drawing	**Yes** recreates the drawing.
Attributes 1 and **Attributes 2** tabs in view properties (**Attributes** and **Shortening** tabs in general arrangement drawings) :	
Scale	Sets the view scale.
Reflected view	Displays load bearing structures, such as columns and beams on a lower floor. **Yes** displays structures with a continuous line, and **No** displays them with a dashed line.
Rotation around \n (in 3D views)	Modify the view angle of 3D views. Enter the values for the angles in y and x directions. Rotation in a drawing view is around the local axis. Value 0.0 for both equals the front view.
Size	**Fit by parts** Tekla Structures fits the view contents in the drawing view frame without leaving any unnecessary space. **Define as distances** The x and y boxes define the view size along the x and y axes of the view. The depth boxes define the depth of the view relative to, and perpendicular to the view plane.

Option	Description
View extension for neighbor parts	Sets the distance from the drawing view to display neighboring parts. For more information, see Show neighbor parts in drawings.
Place	Set the placing for the drawing view to be fixed or free: • **Fixed**: Maintain the views in the same location during updates. • **Free**: Let Tekla Structures find a suitable place for the view during updates. For more information, see Define automatic free or fixed placement for drawing views.
Undeformed	**Yes** undeforms deformed parts and shows the developed (undeformed) shape of the deformed parts in drawing views. For more information, see Undeform deformed parts in drawings.
Shortening	If parts are very long and do not include any details, you can shorten them: • **Cut parts**: **Yes** activates shortening. You can also select to cut **Only in x direction** or **Only in y direction**. • **Minimum cut part length** defines the minimum length of the shortened part to show. • **Space between cut parts** defines the distance between cut parts. • **Cut skew parts**: **Yes** cuts skew parts. For more information, see Shorten parts in drawing views.
Show openings/recess symbol	**Yes** shows symbols in openings and recesses. For more information, see Show part openings and recesses in drawings.

Option	Description
Location by	Select model origin, project base point or any base point defined coordinate system. **Location by** uses the project base point as the default value. • Base point data can be used to set the coordinate system for the view. The base point can be used instead of datum offset. • When the base point is set, level attributes and template attributes in marks provide values in the specific base-point-defined coordinate system. • If you change the project base point Z or the elevation value, the level value will change accordingly when a drawing is opened. • This setting affects level marks and attributes that end with `_BASEPOINT`.
Datum point for elevations	**Specified** uses the entered value. **View plane** measures reference points relative to the view plane. For more information, see Add elevation dimensions.
Show pours in drawing	Yes shows pours in drawings. For more information, see .
Dimension creation method in this view	Clones the dimensions separately for the selected view only. Using this option affects the creation of the dimensions during cloning and re-dimensioning of existing drawings. For more information, see Clone dimensions in selected views only.
Labels tab:	
Text	Defines the view label text. Enter text in fields **A1**-**A5**, or click the ... buttons and select the label mark contents and appearance. For more information about view labels, see

Option	Description
Symbol	Defines a symbol to be used in the view label, and the color, size, and line length of the symbol.
Label position	Defines the vertical and horizontal position of the view label: • For **Vertical**, select **Above** or **Below**. • : Select one of the following for **Horizontal**: **Left aligned to view frame** **Center aligned to view frame** **Right aligned to view frame** **Left aligned to view restriction box** **Center aligned to view restriction box** **Right aligned to view restriction box**
View direction marks	Shows or hides the view direction marks and defines the mark height.
Other options in the view properties options tree:	
Dimensioning	Define view-level dimensions settings for each view separately. For more information on dimensioning settings, see , and .
Filter	Define drawing view filters. For more information, see Create filters.
Neighbor part filter	Define drawing view filters for neighbor parts. For more information, see Create filters.
Protection	Define protected areas to prevent text, marks or dimensions from being placed there. For more information, see .
Marks	Define view-level mark properties. For more information, see Mark properties - Content, General,

Option	Description
	Merging and Appearance tabs (page 734).
Objects	Define view-level properties for the following objects: Parts and neighbor parts (page 759) Bolts (page 764) Surface treatment (page 765) Welds (page 786) Reinforcement and neighbor reinforcement (page 768) Reference objects Grid (page 789) Pour objects and pour breaks
Anchor bolt plan (general arrangement drawings only)	
Show as anchor bolt plan	**Yes** shows the general arrangement drawing as an anchor bolt plan. For more information, see
Enlarged part view scale	Defines the scale used in the enlarged part views.
Create detail views	**Yes** creates separate detail views. If you select **No**, Tekla Structures dimensions the anchor bolts in the enlarged view. Tekla Structures groups similar detail views so that similar details are drawn only once.
Detail view scale	Defines the scale used in the anchor bolt plan detail views.

Section view properties

When you create automatic section views, Tekla Structures creates section views and section marks using the current view and mark properties. You can modify the section view properties in an open drawing.

Option	Description
Attributes tab	

Option	Description
Fit by parts	The **Fit by parts** setting works as an alternative to **Section depth** and **Distance for combining cuts**, and it shows the whole part in the sections.
Section depth	Defines the positive and negative distances of the section view when views are not combined.
Distance for combining cuts	Defines the distance range for combining cut views.
Direction	Define the view direction of the section. The options are: • **Right section** • **Middle section** • **Left sectionLeft section** The available values are **left** or **right**.
Cutting line tab	
Line	Length and offset of the cutting line.
Properties	Color of the cutting line.
Section mark tab	
Text	Defines the text on the section mark. Click the **...** buttons next to the text boxes to open the **Mark Contents** dialog box.
Symbol: **Color**	Color of the section mark symbol.
Left symbol, **Right symbol**	Left and right section mark symbol.
Size	Size of the section mark left and right symbol.
x/y	Offset of the section mark left and right symbol.
Start number or letter of section view and symbol label	Defines the letter or number used in the section view label or in the section symbol label. You can enter any number starting from 1 or any letter A - Z or a - z (also shown in uppercase in the label). If you use letter, and the entered string is longer than one letter, only the first letter is shown in the view label and section symbol label. If you use numbers, all entered numbers are shown. The start number changes

Option	Description
	only if you change it in the drawing properties before creating a drawing, and if you change the option in an existing drawing and recreate the drawing, in which case section view and symbol labels for all automatically included section views and all new section views will change.

See also

View properties in drawings (page 676)

Dimension and dimensioning properties in drawings

Dimension and dimensioning properties define what the dimensions look like, and which formats, units etc. are used, and what is being dimensioned and how.

Click the following links to find out more about dimension properties:

- Dimension properties in drawings (page 684)
- Rebar dimension mark properties (page 700)

Click the following links to find out more about dimensioning properties in view-level dimensioning:

-
- Dimensioning properties in drawings (Integrated dimensioning) (page 709)

Click the following link to find out more about dimensioning properties in GA drawings:

- General arrangement drawing dimensioning properties (page 729)

Dimension properties in drawings

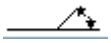
Use the **Dimension Properties** dialog box to view and modify the dimension properties in drawings.

- To open the **Dimension Properties** dialog box, open the drawing, go to the **Drawing** tab and click **Properties** --> **Dimension** .
- To open the object level **Dimension Properties**, double-click a dimension in an open drawing.

- To open **Dimension Properties** dialog box for general arrangement drawings before creating a drawing: On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** , and then click **Dimension...**

General tab

Option	Description
Dimension types	
Straight	Sets the dimension type for straight dimensions.
	Relative: Point to point dimensions.
	Absolute: Dimensions from a common start point.
	Relative and absolute: Combination of point to point and common start point.
	US absolute: Dimensions from a common start point, which include a running dimension mark (RD).
	US absolute 2: Similar to **US absolute**, but it changes short dimensions to relative.
	Absolute plus short relatives: Similar to **Absolute**, but it changes short dimensions to relative. Also called internal absolute. This option may show both dimensions, but it does not show relative dimensions when dimensions are long. This option shows the absolute dimensions inside the dimension lines.
	Absolute plus all relatives above the absolutes: Similar to **Relative and absolute**, but it places the relative dimensions above the absolute.
	Elevation: Creates an elevation dimension at a picked point. This type is only available in the dimension properties of manual dimensions in drawing mode.
In x direction	As above, but overrides the straight setting for horizontal dimensions. If you use the blank option, Tekla

Option	Description
	Structures uses the **Straight** option settings. The x direction usually means dimensions parallel to the drawing x axis.
Angle	Defines the appearance of angle dimensions.
	Shows the angle dimensions in degrees on side.
	Shows the angle dimensions in degrees at angle vertex.
	Shows the angle dimensions using a triangle. You can also set the **Triangle base length** to control the base dimension shown for bevel dimensions.
	Shows the angle dimensions using a triangle with degrees.
Triangle base length	The base length of a triangle.
Curved	Defines whether to use angle units or distance units for curved dimensions.
Short extension line	Defines whether Tekla Structures should create extension lines all of the same length or use the short extension line automatically if the dimension line falls on a grid line. If you want to exaggerate dimensions, you need to set this option to **No**.
Dimension format	
Units	Defines the units used in dimensioning. See the table below for details.
Precision	Defines dimension precision: rounding, imperial units. See the table below for details.
Format	Defines the dimension format: the number of decimals and their appearance. See the table below for details.
Use grouping	Defines whether long dimension values are grouped.

Option	Description
Combine equal dimensions	Combines equal dimensions. The choices are **Off**, **3*60** or **3*60=180**. The accuracy of combining equal dimensions is 0.1.
Minimum number to combine	Defines the minimum number of dimensions to combine.
Dimension grouping	
Update grouping when model changes	**Yes** updates dimension grouping automatically when the model changes.
Placing	
Dimension lines spacing	Defines the space between parallel dimension lines. In manually created dimensions, this setting works only if dimension **Placing** is set to **free**, see below.
Short dimensions	Defines the text location of short dimensions: between or outside the dimension lines.
Place...	Opens the **Dimension Placing** dialog box. **Placing** is the method used for placing dimensions. The choices are: • **free** lets Tekla Structures to decide the location and direction of the dimension based on the **Direction** settings. • **fixed** allows you to place the dimension at any point. **Search margin** is the empty margin that you want to have around the dimension. **Minimum distance** is the closest distance Tekla Structures uses to search for an empty space for a dimension. **Direction** defines the side of the dimensioned object on which Tekla Structures places the dimensions. For more information about dimension placement properties, see

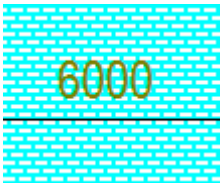
Option	Description
	Placement properties for dimensions, marks, notes, texts and symbols (page 784).

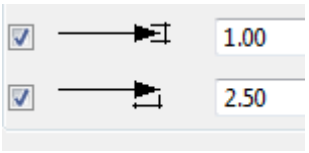
Format, precision and units

Option	Integer value	Notes
Format		
###	0	
###[.#]	1	
###.#	2	
###[.##]	3	
###.##	4	
###[.###]	5	
###.###	6	
### #/#	7	
## # /##.## #	8	This option is available only for straight dimensions.
Precision		
0.00	1	For defining precision with rounding. For example, with precision 0.33 the actual dimension 50.40 is shown as 50.33.
0.50	2	
0.33	3	
0.25	4	
1/8	5	For imperial units
1/16	6	
1/32	7	
1/10	8	For defining precision without rounding
1/100	9	
1/1000	10	
Units		
automatic		Uses the units defined in the model
mm		millimeters
cm		centimeters
m		meters

Option	Integer value	Notes
foot - inch		feet and inches Inches are converted into integer feet, and the remaining inches are shown in inches.
cm / m		centimeters and meters Dimensions under 100 cm are shown in centimeters, and dimensions above 100 cm are shown in meters. Millimeters are shown as superscript.
inch		inches
feet		feet

Appearance tab

Option	Description
Text	
Color	The color of the dimension mark text. This controls the line weight in printed drawings.
Background mask	Select **Opaque** to hide the area of the drawing that is covered by the dimension mark. Select **Transparent** to show the drawing content in the dimension mark background so that drawing linework is visible. 
Height	Controls the height of the text used in the dimension marks in the drawing.

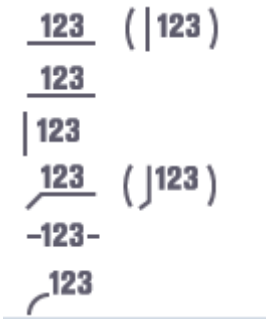
Option	Description
Font	Controls which font is used in the dimension mark.
Frame	Defines a frame for the dimension mark.
Place	Defines how the dimension mark is placed relative to the dimension line.
Line, Arrow	
Color	Controls the color of the dimension line. The color controls the line weight in printed drawings.
Arrow shape	Controls the type of mark used with the dimension line. You can also create arrow types of your own, see .
US Absolute dimensions	Controls the type of mark used with the dimension line in US Absolute dimensions. This setting is only available in GA drawing dimension properties.
Level dimensions	Controls the type of mark used with the dimension line in level dimensions. This setting is only available in GA drawing dimension properties.
	Sets the height and the length of the arrowhead.

Marks and Tags tabs

Option	Description
Marks tab	
Prefix	Displays the defined value before the numeric value of the dimension. Enter text or click **...** to select elements. The prefix value cannot be plain numbers, and cannot end with a number.

Option	Description
Visibility of numeric value	Defines whether the numeric value of the dimension is visible or hidden. If you hide the numeric value of the dimension, the prefix and postfix text will still be shown.
Postfix	Displays the defined value after the numeric value of the dimension. Enter text or click **...** to select elements. The postfix cannot be plain numbers, and the it cannot start with a number when the numeric value of the dimension is visible.
... buttons	Define dimension mark contents by adding elements. You can also modify the mark appearance. **Frame around elements**: **Type** and **Color** define the element frame type and color individually for each element. **< Add frame** allows you to add frames around elements. **Font**: **Color**, **Height** and **Font** define the font type, color, and height used in element texts individually for each element. Clicking **Select...** gives more font choices. **Units**: **Units** and **Format** allow you to set the unit and format for length elements. Unit settings can be modified only when a length element is selected. You can also use template attributes in the **User-defined attribute** element. In marks, you cannot use template attributes such as `MODEL_TOTAL` that refer to the whole model. Marks only check the information from the object in the drawing and not from the whole model.
Plate side marks	**Specified** creates plate side marks to plate dimension using the specified color, size and offset. Offset means

Option	Description
	the distance of the mark from the dimension line. The option **Automatic** is available only in intelligent drawings, that is when the advanced option `XS_INTELLIGENT_DRAWING_ALLOWED` is set to `TRUE`.
Exaggeration	**Specified** exaggerates narrow dimensions. Select the **Direction**: **Left / Down**, **Right / Up** or **Both**. Set the **Origin**, **Width**, **Position** and **Height**.
Tags tab	
Tags area	Add tags to dimensions: You can enter text in the tag box or add elements by clicking the **...** When you click the **...** button next to a tag box, a dialog box for the particular tag is displayed and you can define dimension tag contents by adding elements. You can also modify the tag element appearance. • **Rotation** --> **Perpendicular to dimension line** rotates the tag. • **Rotation** --> **Parallel to dimension line** does not rotate the tag. This is the default value. • **Frame around elements**: **Type** and **Color** define the element frame type and color individually for each element. • **< Add frame** allows you to add frames around elements. **Font**: **Color**, **Height** and **Font** define the font type, color, and height used in element texts individually for each element. Clicking **Select...** gives more font choices. • **Units**: **Units** and **Format** allow you to set the unit and format for length elements. Unit settings can

Option	Description
	be modified only when a length element is selected. • You can also use template attributes in the **User-defined attribute** element. In tags, you cannot use template attributes such as `MODEL_TOTAL` that refer to the whole model. Tags only check the information from the object in the drawing, and not from the whole model.
Include part count in the tag	Select the tag positions for which you want to include the part count in the tag.
Exclude parts according to filter	Select a drawing view filter that removes desired content from the tag.
Curved Dimension Tag Type	Select a tag type for a curved reinforcing bar dimension line. The tag type controls how the tags are aligned to the dimension. 

Dimension properties - Units, precision and format

Use the **General** tab of the **Dimension properties** dialog box to view and modify options related to dimension format, unit and precision.

The integer values are provided for situations when you need to use the value as a value for an advanced option.

Option	Integer value	Notes
Format		
###	0	
###[.#]	1	

Option	Integer value	Notes
###.#	2	
###[.##]	3	
###.##	4	
###[.###]	5	
###.###	6	
### #/#	7	
## # /##.## #	8	This option is available only for straight dimensions.
Precision		
0.00	1	For defining precision with rounding. For example, with precision 0.33 the actual dimension 50.40 is shown as 50.33.
0.50	2	
0.33	3	
0.25	4	
1/8	5	For imperial units
1/16	6	
1/32	7	
1/10	8	For defining precision without rounding
1/100	9	
1/1000	10	
Units		
automatic		Uses the units defined in the model
mm		millimeters
cm		centimeters
m		meters
foot - inch		feet and inches Inches are converted into integer feet, and the remaining inches are shown in inches.
cm / m		centimeters and meters Dimensions under 100 cm are shown in centimeters, and dimensions above 100 cm are shown in meters.

Option	Integer value	Notes
		Millimeters are shown as superscript.
inch		inches
feet		feet

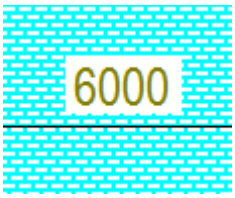
See also

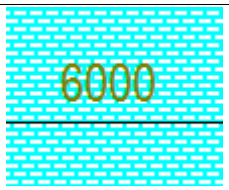
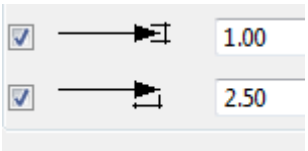
Dimension properties in drawings (page 684)

Dimension properties - Appearance tab

Use the **Appearance** tab in the **Dimension Properties** dialog box to view and modify the settings affecting the dimension appearance.

- To open the applied **Dimension Properties** dialog box, open the drawing, go to the **Drawing** tab and click **Properties** --> **Dimension**.
- To open the object level **Dimension Properties** dialog box, double-click a dimension in an open drawing.
- To open **Dimension Properties** dialog box for general arrangement drawings before creating a drawing: On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing**, and then click **Dimension**.

Option	Description
Text	
Color	The color of the dimension mark text. This controls the line weight in printed drawings.
Background mask	Select **Opaque** to hide the area of the drawing that is covered by the dimension mark.  Select **Transparent** to show the drawing content in the dimension mark background so that drawing linework is visible.

Option	Description
	
Height	Controls the height of the text used in the dimension marks in the drawing.
Font	Controls which font is used in the dimension mark.
Frame	Defines a frame for the dimension mark.
Place	Defines how the dimension mark is placed relative to the dimension line.
Line, Arrow	
Color	Controls the color of the dimension line. The color controls the line weight in printed drawings.
Arrow shape	Controls the type of mark used with the dimension line. You can also create arrow types of your own, see .
US Absolute dimensions	Controls the type of mark used with the dimension line in US Absolute dimensions. This setting is only available in GA drawing dimension properties.
Level dimensions	Controls the type of mark used with the dimension line in level dimensions. This setting is only available in GA drawing dimension properties.
	Sets the height and the length of the arrowhead.

See also

Dimension properties in drawings (page 684)

Dimension properties - Marks and Tags tabs

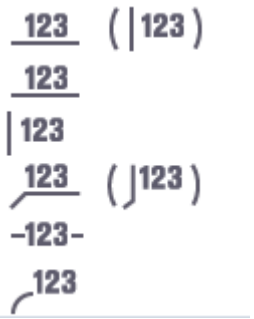
Use the **Marks** and **Tags** tabs in the **Dimension Properties** dialog box to view and modify the contents of dimension marks and tags in an open drawing.

- To open the drawing level **Dimension Properties** dialog box, open the drawing, go to the **Drawing** tab and click **Properties** --> **Dimension** .
- To open the object level **Dimension Properties**, double-click a dimension in an open drawing.
- To open **General - Dimension Properties** dialog box for general arrangement drawings before creating a drawing: On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** , and then click **Dimensioning...**

Option	Description
Marks tab	
Prefix	Displays the defined value before the numeric value of the dimension. Enter text or click **...** to select elements. The prefix value cannot be plain numbers, and cannot end with a number.
Visibility of numeric value	Defines whether the numeric value of the dimension is visible or hidden. If you hide the numeric value of the dimension, the prefix and postfix text will still be shown.
Postfix	Displays the defined value after the numeric value of the dimension. Enter text or click **...** to select elements. The postfix cannot be plain numbers, and the it cannot start with a number when the numeric value of the dimension is visible.
... buttons	Define dimension mark contents by adding elements. You can also modify the mark appearance. **Frame around elements**: **Type** and **Color** define the element frame type and color individually for each element. **< Add frame** allows you to add frames around elements. **Font**: **Color**,

Option	Description
	Height and **Font** define the font type, color, and height used in element texts individually for each element. Clicking **Select...** gives more font choices. **Units**: **Units** and **Format** allow you to set the unit and format for length elements. Unit settings can be modified only when a length element is selected. You can also use template attributes in the **User-defined attribute** element. In marks, you cannot use template attributes such as `MODEL_TOTAL` that refer to the whole model. Marks only check the information from the object in the drawing and not from the whole model.
Plate side marks	**Specified** creates plate side marks to plate dimension using the specified color, size and offset. Offset means the distance of the mark from the dimension line. The option **Automatic** is available only in intelligent drawings, that is when the advanced option `XS_INTELLIGENT_DRAWING_ALLOWED` is set to `TRUE`.
Exaggeration	**Specified** exaggerates narrow dimensions. Select the **Direction**: **Left / Down**, **Right / Up** or **Both**. Set the **Origin**, **Width**, **Position** and **Height**.
Tags tab	
Tags area	Add tags to dimensions: You can enter text in the tag box or add elements by clicking the **...** When you click the **...** button next to a tag box, a dialog box for the particular tag is displayed and you can define dimension tag contents by adding

Option	Description
	elements. You can also modify the tag element appearance. • **Rotation** --> **Perpendicular to dimension line** rotates the tag. • **Rotation** --> **Parallel to dimension line** does not rotate the tag. This is the default value. • **Frame around elements**: **Type** and **Color** define the element frame type and color individually for each element. • **< Add frame** allows you to add frames around elements. **Font**: **Color**, **Height** and **Font** define the font type, color, and height used in element texts individually for each element. Clicking **Select...** gives more font choices. • **Units**: **Units** and **Format** allow you to set the unit and format for length elements. Unit settings can be modified only when a length element is selected. • You can also use template attributes in the **User-defined attribute** element. In tags, you cannot use template attributes such as `MODEL_TOTAL` that refer to the whole model. Tags only check the information from the object in the drawing, and not from the whole model.
Include part count in the tag	Select the tag positions for which you want to include the part count in the tag.
Exclude parts according to filter	Select a drawing view filter that removes desired content from the tag.
Curved Dimension Tag Type	Select a tag type for a curved reinforcing bar dimension line. The

Option	Description
	tag type controls how the tags are aligned to the dimension. 

Rebar dimension mark properties in drawings

Use the **Rebar dimension mark properties** dialog box to define what rebar dimensions look like and which formats, units etc. are used.

- To open the **Rebar dimension mark properties** dialog box, open the drawing, go to the **Drawing** tab and click **Properties** --> **Rebar dimension mark**.

  For details, see Add dimensions to reinforcement.

Rebar tab

Use the **Rebar** tab in the **Rebar dimension mark properties** dialog box to modify the style of the rebar dimension mark in an open drawing.

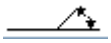
Option	Description
Dimension to	Specifies whether dimensions are created between **All rebars** or between the **Start and end rebars** only.
Part edge	Adds closing dimensions to the edge of the part in reinforcing bar group dimensions. This setting is only available if **Dimension to** is set to **All rebars**.
Subgrouping	Specifies whether subgroups are created for different rebar spacings within a rebar object so that dimension marks can display the subgroup properties. **Subgrouping** is only available when **Dimension to** is set to **All rebars** and is disabled for **Start and end rebars**.

Option	Description
Extension line to visible rebar	Controls the display of extension lines to the visible rebars.
Tapered dimension type	Specifies whether skewed dimensions have skewed or horizontal representation .
Curved dimension type	Specifies whether curved dimensions have curved or horizontal representation .

General tab

Use the **General** tab in the **Rebar dimension mark properties** dialog box to view and modify the format, type, unit, precision, extension line, grouping and placing settings of the dimensions.

Option	Description
Dimension types	
Straight	Sets the dimension type for straight dimensions.
	Relative: Point to point dimensions.
	Absolute: Dimensions from a common start point.
	Relative and absolute: Combination of point to point and common start point.
	US absolute: Dimensions from a common start point, which include a running dimension mark (RD).
	US absolute 2: Similar to **US absolute**, but it changes short dimensions to relative.
	Absolute plus short relatives: Similar to **Absolute**, but it changes short dimensions to relative. Also called internal absolute. This option may show both dimensions, but it does not show relative dimensions when dimensions are long. This option shows the absolute dimensions inside the dimension lines.

Option	Description
	Absolute plus all relatives above the absolutes: Similar to **Relative and absolute**, but it places the relative dimensions above the absolute.
	Elevation: Creates an elevation dimension at a picked point. This type is only available in the dimension properties of manual dimensions in drawing mode.
In x direction	As above, but overrides the straight setting for horizontal dimensions. If you use the blank option, Tekla Structures uses the **Straight** option settings. The x direction usually means dimensions parallel to the drawing x axis.
Angle	Defines the appearance of angle dimensions.
	Shows the angle dimensions in degrees on side.
	Shows the angle dimensions in degrees at angle vertex.
	Shows the angle dimensions using a triangle. You can also set the **Triangle base length** to control the base dimension shown for bevel dimensions.
	Shows the angle dimensions using a triangle with degrees.
Triangle base length	The base length of a triangle.
Short extension line	Defines whether Tekla Structures should create extension lines all of the same length or use the short extension line automatically if the dimension line falls on a grid line. If you want to exaggerate dimensions, you need to set this option to **No**.
Dimension format	
Units	Defines the units used in dimensioning. **automatic**

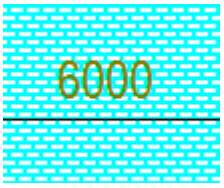
Option	Description
	Uses the units defined in the model. **mm** (millimeters) **cm** (centimeters) **m** (meters) **foot - inch** (feet and inches) Inches are converted into integer feet, and the remaining inches are shown in inches. **cm / m** (centimeters and meters) Dimensions under 100 cm are shown in centimeters, and dimensions above 100 cm are shown in meters. Millimeters are shown as superscript. **inch** (inches) **feet** (feet)
Precision	Defines dimension precision: rounding, imperial units. The following are for defining precision with rounding. For example, with precision 0.33 the actual dimension 50.40 is shown as 50.33. **0.00** **0.50** **0.33** **0.25** The following are for imperial units: **1/8** **1/16** **1/32** The following are for defining precision without rounding: **1/100** **1/1000** **1/10**

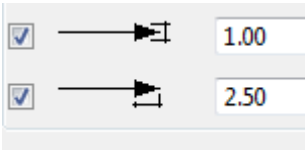
Option	Description
Format	Defines the dimension format: the number of decimals and their appearance. **###** **###[.#]** **###.#** **###[.##]** **###.##** **###[.###]** **###.###** **### #/#** The following is only available for straight dimensions: **## #/##.## #**
Use grouping	Defines whether long dimension values are grouped.
Combine equal dimensions	Combines equal dimensions. The choices are **Off**, **3*60** or **3*60=180**. The accuracy of combining equal dimensions is 0.1.
Minimum number to combine	Defines the minimum number of dimensions to combine.
Placing	
Dimension lines spacing	Defines the space between parallel dimension lines. In manually created dimensions, this setting works only if dimension **Placing** is set to **free**, see below.
Short dimensions	Defines the text location of short dimensions: between or outside the dimension lines.
Place...	Opens the **Dimension Placing** dialog box. **Placing** is the method used for placing dimensions. The choices are: • **free** lets Tekla Structures to decide the location and direction of the dimension based on the **Direction** settings.

Option	Description
	• **fixed** (default) allows you to place the dimension at any point. **Search margin** is the empty margin that you want to have around the dimension. **Minimum distance** is the closest distance Tekla Structures uses to search for an empty space for a dimension. **Direction** defines the side of the dimensioned object on which Tekla Structures places the dimensions.

Appearance tab

Use the **Appearance** tab in the **Rebar dimension mark properties** dialog box to view and modify the settings affecting the dimension appearance.

Option	Description
Text	
Color	The color of the dimension mark text. This controls the line weight in printed drawings.
Background mask	Select **Opaque** to hide the area of the drawing that is covered by the dimension mark. Select **Transparent** to show the drawing content in the dimension mark background so that drawing linework is visible. 
Height	Controls the height of the text used in the dimension marks in the drawing.

Option	Description
Font	Controls which font is used in the dimension mark.
Frame	Defines a frame for the dimension mark.
Place	Defines how the dimension mark is placed relative to the dimension line.
Line, Arrow	
Color	Controls the color of the dimension line. The color controls the line weight in printed drawings.
Arrow shape	Controls the type of mark used with the dimension line. You can also create arrow types of your own, see .
	Sets the height and the length of the arrowhead.

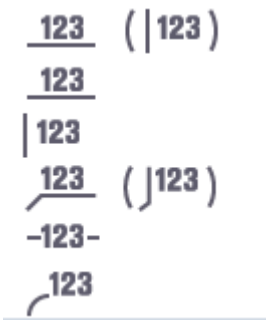
Marks and Tags tabs

Use the **Marks** and **Tags** tabs in the **Rebar dimension mark properties** dialog box to view and modify the contents of rebar dimension marks and tags in an open drawing.

Option	Description
Marks tab	
Prefix	Displays the defined value before the numeric value of the dimension. Enter text or click **...** to select elements. The prefix value cannot be plain numbers, and cannot end with a number.
Visibility of numeric value	Defines whether the numeric value of the dimension is visible or hidden. If you hide the numeric value of the dimension, the prefix and postfix text will still be shown.
Postfix	Displays the defined value after the numeric value of the dimension.

Option	Description
	Enter text or click **...** to select elements. The postfix cannot be plain numbers, and the it cannot start with a number when the numeric value of the dimension is visible.
... buttons	Define dimension mark contents by adding elements. You can also modify the mark appearance. **Frame around elements**: **Type** and **Color** define the element frame type and color individually for each element. **< Add frame** allows you to add frames around elements. **Font**: **Color**, **Height** and **Font** define the font type, color, and height used in element texts individually for each element. Clicking **Select...** gives more font choices. **Units**: **Units** and **Format** allow you to set the unit and format for length elements. Unit settings can be modified only when a length element is selected. You can also use template attributes in the **User-defined attribute** element. In marks, you cannot use template attributes such as `MODEL_TOTAL` that refer to the whole model. Marks only check the information from the object in the drawing and not from the whole model.
Plate side marks	**Specified** creates plate side marks to plate dimension using the specified color, size and offset. Offset means the distance of the mark from the dimension line. The option **Automatic** is available only in intelligent drawings, that is when the advanced option `XS_INTELLIGENT_DRAWING_ALLOWED` is set to `TRUE`.

Option	Description
Exaggeration	**Specified** exaggerates narrow dimensions. Select the **Direction**: **Left / Down**, **Right / Up** or **Both**. Set the **Origin**, **Width**, **Position** and **Height**.
Tags tab	
Tags area	Add tags to dimensions: You can enter text in the tag box or add elements by clicking the **...** When you click the **...** button next to a tag box, a dialog box for the particular tag is displayed and you can define dimension tag contents by adding elements. You can also modify the tag element appearance. • **Rotation** --> **Perpendicular to dimension line** rotates the tag. • **Rotation** --> **Parallel to dimension line** does not rotate the tag. This is the default value. • **Frame around elements**: **Type** and **Color** define the element frame type and color individually for each element. • **< Add frame** allows you to add frames around elements. **Font**: **Color**, **Height** and **Font** define the font type, color, and height used in element texts individually for each element. Clicking **Select...** gives more font choices. • **Units**: **Units** and **Format** allow you to set the unit and format for length elements. Unit settings can be modified only when a length element is selected. • You can also use template attributes in the **User-defined attribute** element. In tags, you cannot use template attributes such as `MODEL_TOTAL`

Option	Description
	that refer to the whole model. Tags only check the information from the object in the drawing, and not from the whole model.
Include part count in the tag	Select the tag positions for which you want to include the part count in the tag.
Exclude parts according to filter	Select a drawing view filter that removes desired content from the tag.
Curved Dimension Tag Type	Select a tag type for a curved reinforcing bar dimension line. The tag type controls how the tags are aligned to the dimension. 

Dimensioning properties in drawings (Integrated dimensioning)

Use the **General** tab in the **Dimension Properties** dialog box to view and modify the general settings of the dimensions. This dialog box is displayed if you use **Integrated dimensions** dimensioning type in dimensioning.

General tab

Option	Description
Dimensioning type	**Standard** is used for almost all dimensioning. **Truss** meets the specific requirements needed to dimension truss drawings. It dimensions the position and length of the diagonals. The dimensioning is done only if the diagonals are secondary parts that are welded to upper and lower chords, which are main parts and not welded to any parts. If the truss

Option	Description
	welding is done some other way, standard dimensioning is used.
Minimize	**Yes** minimizes the number of views that Tekla Structures creates. Also check the settings in the drawing **View Properties** dialog box.
Combine dimensions	Combines several single dimensions into one dimension line. In **Options**, select the combination level. The larger the number, the more Tekla Structures combines dimensions. Option **4.5** uses a combination of option **5** for main part and option **4** for secondary parts. The **Distance** means the distance within which Tekla Structures combines internal dimensions. If the distance between two details is less than the defined **Min distance**, Tekla Structures combines the dimensions.
Close dimensions	Close dimensions completes dimension lines to include the entire part. **No** does not close dimensions **In X** only closes dimensions in the x direction and leaves others open **All** closes all dimensions This setting is not relevant to profile shape dimensions.
Close dimensions: **Short dimensions**	**Yes** closes short dimensions. With **No**, the open dimension is the middle one rather than the short end dimension. When you leave short dimensions open, Tekla Structures leaves out the longer dimension line in dimension lines that contain two dimensions. If dimension lines contain three dimensions, Tekla Structures leaves

Option	Description
	out the middle one. This option does not affect dimension lines with more than three dimensions.
Placing: **Forward offset**	Forward offset defines the distance Tekla Structures uses to search for the base point of a dimension. If Tekla Structures does not find a base point (corner) within the **Forward offset** search distance, it uses an edge point. The **Centered bolt** setting affects on how the dimension is displayed.
Necessary internal dimensioning: **Recognizable distance**	Recognizable distance defines the limit for dimensioning asymmetry in secondary parts. In some cases, it is important to look at the asymmetrical relationship of parts, so that an asymmetric secondary part is correctly connected to a main part. You can use **Recognizable distance** to have asymmetry reflected in dimensioning. If the asymmetry is smaller than the distance you enter here, Tekla Structures represents it using a dimension.
Part mark on dimension line	**None** creates no part mark on the dimension line. **On assembly overall dimension** creates a part mark on the assembly overall dimension line. **Between extreme bolts** creates a part mark on the dimension line between the outermost bolts.
Main dimension position	Defines the side on which Tekla Structures places the **Assembly overall dimensions**, **Main part work points**, and **Knock off dimensions**. **Auto** treats the main dimensions in the same way as other dimensions. **Above** places the main dimensions above the part (or to the left for vertical parts).

Option	Description
	Below places the main dimensions below the part. **Above** places the skew position dimensions of the main part below the part and **Below** places them above.
Grid dimensions	Creates grid dimensions. The values are: **None**, **Individual spans**, **Overall** or **Individual spans and overall**.
Grid dimension position	Sets the grid dimension position. The values are: **Main view - above** **Main view - below** **Top view - above** **Top view - below** **Bottom view - above** **Bottom view - below** **Back view - above** **Back view - below** **All views - above** **All views - below**.
Dimension properties	
Straight dimensions	Sets the dimension type for straight dimensions using the settings in the properties file you select.
In X direction	Otherwise same settings as in straight dimensions but overrides the straight setting for horizontal dimensions. If you use the blank option, Tekla Structures uses the **Straight dimensions** option settings. The x direction usually means dimensions parallel to the drawing x axis.
Arrow shape: **US Absolute dimensions**	Controls the type of mark used with the dimension line in US Absolute dimensions. US Absolute dimensions are dimension types **US Absolute** and **US Absolute 2** that can be defined in Dimension properties (page 684). If

Option	Description
	the advanced option XS_USE_USABSOLUTE_ARROW_TYPE_FOR_ABSOLUTE_DIMENSIONS (page 487) is set to `TRUE`, the arrow shape is overridden for other types of absolute dimensions as well.
Arrow shape: **Level dimensions**	Controls the type of mark used with the dimension line in level dimensions.
Angle and radius dimensions	Sets the dimension type for angle dimensions using the settings in the properties file you select.
Check dimensions	Sets the dimension type for check dimensions using the settings in the properties file you select.

Position dimensions tab

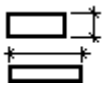
Option	Description
Position bolts to **Position parts to**	Controls from where Tekla Structures creates the part/bolt position dimensions. **None** creates no position dimensions. **Main part** creates the dimensions from the main part reference line. **Working points** creates the dimensions between the work points, such as the intersections of main and neighbor part reference lines.
Embedded objects	Creates position dimensions for locating embedded objects in cast unit drawings. The embedded objects are custom components attached to the cast unit. **As secondary objects** dimensions embedded objects in cast unit drawings the same way as secondary parts. **By reference point** dimensions embedded objects to their reference point, which is the origin of the custom component.

Option	Description
Secondary part	Creates dimensions to bolt holes or edges of the secondary part. **None** creates no position dimensions for secondary parts. **By bolt** dimensions bolt hole locations in the secondary parts. **By part** dimensions edges of secondary parts. **By both** dimensions bolt hole locations and edges of the secondary part.
Secondary part dimension direction	Aligns dimensions with the main or neighboring part. Only for skewed clip angles or shear plates.
Position from	Sets the start point for running dimensions. Only for skewed clip angles or shear plates bolted to a neighboring part.
Reversed direction for running dimensions	**Yes** changes the direction of running dimensions. With this option, you can set the zero point to the end of a member rather than to the start.
Main part bolt position	**On** creates dimensions to the bolt hole locations in the main part.
Main part skew position	**Yes** creates horizontal and vertical check dimensions representing the skew position of a brace. Created between the work points of the main part. Skew check dimensions are located in the front view. Their location depends on the **Main dimension position** setting selected on the **General** tab. When the main dimensions are above the part, the skew dimensions are below, and vice versa.
Skew position	Defines how Tekla Structures dimensions skewed secondary part positions. **None** creates no dimensions for skewed secondary part positions.

Option	Description
	angle creates an angle dimension for the secondary part. **Dimensions** creates dimensions for the skewed position of the secondary part. **Both** creates both the angle and the dimensions.
Centered part	Controls the dimensions of centrally-placed parts. These settings have an effect only if position dimensions are created. **Internal** dimensions the centered parts overall size. **Position** dimensions the part to the main part center lines. **None** creates no centered part dimensions.
Centered bolt	Controls the dimensions of centrally-placed bolt groups. **Internal** dimensions the centered bolts spread. **Position** dimensions the bolts to the main part center-lines. **Centered bolt** overrides the **Secondary part bolt internal** option for centered bolts. This only applies to bolts located centrally on the part.
Elevation dimensions	**On** creates elevation dimensions.
Combine equal dimensions	Combines equal dimensions. The choices are **Off**, **3*60** or **3*60=180**. The accuracy of combining equal dimensions is 0.1.
Minimum number to combine	Defines the minimum number of dimensions to combine.

Part dimensions tab

Option	Description
Internal	Creates internal dimensions for secondary parts connected to the main part. **None** creates dimensions for secondary parts. **Necessary** creates only the dimensions that are necessary for assembling the parts. **All** creates all dimensions for the secondary parts.
Main part overall dimensions	**Once** creates one overall dimension for the main part. **All** creates overall dimensions for main parts in all views. **None** creates no overall dimensions for the main part. The **Assembly part overall dimensions** settings have some effect on these options.
Assembly part overall dimensions	**Length only** creates overall dimensions for an entire assembly or cast unit in the x direction only. **All dimensions** creates overall dimensions for an assembly or cast unit in all directions. **Off** creates no overall dimensions for an assembly or a cast unit.
Main part work points	**On** creates a check dimension between the outermost work points.
Main part shape (**Shape dimensions**)	**On** creates dimensions to show the shape of a main part. By default, Tekla Structures automatically draws shape dimensions on both ends of a beam, even if the ends are symmetrical.

Option	Description
Main part radius dimensions (**Radius dimensions**)	**On** creates radius dimensions for curved chamfers and round holes in the main part. This option is available only if you have set **Main part shape** dimensions to **On**. Note that this option does not create radius dimensions for curved beams or polybeams with chamfers.
Bevel dimensions	**On** creates linear dimensions of a bevel.
Bevel angle	Creates an angle dimension and defines which side of the bevel to dimension. The choices are **None**, **Angle of cut** and **Angle of beam**.
Knock off dimensions	**On** creates check dimensions from the edge of the main part to the work point.
Preferred dim side	Sets the preferred view (front or side) for part dimensions. 
From the nearest floor level to part	**On** creates dimensions indicating the distance from the closest floor level to the bottom and/or top of the parts.
From grid to part centerline	**On** creates dimensions showing the offset of a part from the grid to the part center line.
From grid to part ends	**On** creates dimensions showing the offset of a part from the grid to the near or far end of the part.

Bolt dimensions tab

Option	Description
Main part bolt internal dimensions	Creates internal dimensions for bolt groups in the main part. **None** creates none of the internal bolt dimensions.

Option	Description
	Internal creates bolt group internal dimensions (distances between bolts). **All** creates edge distance and bolt group internal dimensions. Edge distance is the dimension from the outermost bolt to the part edge.
Main part bolt internal dimensions: **Skewed bolt group**	Indicates whether the dimensions are parallel to the part or the bolt group. The choices are **No dimensions**, **In part direction**, and **In bolt group direction**.
Secondary part bolt internal dimensions	Creates internal dimensions for bolt groups in the secondary part. The choices are **None**, **Necessary**, **Internal**, and **All**.
Secondary part bolt internal dimensions: **Skewed bolt group**	Aligns the bolt dimensions with the secondary part or bolt group. The choices are **In part direction**, **No dimensions**, and **In bolt group direction**
Distance between extreme bolts: **Extreme bolts**	Creates check dimension between the outermost bolts. The choices are **None**, **Main part**, and **Assembly**.
Distance between extreme bolts: **Extreme bolts to work points**	Creates check dimensions from the outermost bolts to the work points. **Yes** creates the check dimensions.
Preferred dim side	Sets the preferred view (front or side) for bolt dimensions.
Combine bolt dimensions	Sets the format of the combined bolt group internal dimensions. You can combine bolt group internal dimensions and display them in the format **3*60** or **3*60=180**, or have single dimensions.
Minimum number to combine	Defines the minimum number of dimensions to combine.

Dimension grouping tab

Option	Description
Activate dimension grouping	Selects the objects for grouping.
Parts	Groups according to parts.
Bolts	Groups according to bolts.
Components	Groups according to components.
Cuts/Shapes	Groups according to cuts or shapes.
Automatic tagging	Defines how to display information in a dimension line.
Display tags	Displays tags.
Include part count in the tag	Includes part count in the tag.
Do not display marks\nfor the grouped items	Does not display part marks for grouped items.
Available elements	Elements available to define identical conditions.
Add >	Adds items to **Selected elements** list.
Remove	Removes items from **Selected elements** list.
Move up	Moves the element higher in the list.
Move down	Moves the element lower in the list.
Update grouping\nwhen model \nchanges	**Yes** updates dimension grouping automatically when the model changes.

Sub-assemblies tab

Option	Description
Dimension parts inside sub-assemblies	Defines whether to dimension parts inside sub-assemblies. **Yes** creates internal dimensions for parts inside sub-assemblies. **No** does not create internal dimensions for parts inside sub-assemblies.
Measure sub-assembly position from	Defines the position from which the sub-assembly is measured. **None** does not measure the sub-assembly position. **Bolt** measures the position of the sub-assembly from the bolts. If bolts are not included in the sub-assembly,

Option	Description
	or if it is not possible to measure the position from bolts, Tekla Structures measures the sub-assembly position from the reference point. **Extrema points** measures the position of the sub-assembly from the bounding box of the sub-assembly. **Reference point** measures the position of the sub-assembly from the reference point.

Reinforcement dimensions tab

Option	Description
Dimensions for reinforcing bar groups	**On** creates dimensions for reinforcing bar groups. This also activates the other selections on this tab.
Dimension mark settings	Sets the mark type, which can be dimension line, dimension mark, tagged dimension mark, or variants of these typical styles. The Default environment contains three predefined settings files: `rebar_dimension_line`, `rebar_dimension_mark`, and `rebar_tagged_dimension_mark` Clicking the **...** button opens the dimension properties, and you can view the settings file and also change the settings, if needed.

Dimensioning properties - Position dimensions tab (Integrated dimensioning)

Use the **Position dimensions** tab in the **Dimensioning Properties** dialog box to view and modify the settings for the position dimensions in single-part, assembly and cast unit drawings.

Note that the contents of the dialog box vary depending on the drawing type, and all the options listed below are not available for all drawing types. This dialog box is displayed if you use **Integrated dimensions** dimensioning type in dimensioning.

Option	Description
Position bolts to **Position parts to**	Controls from where Tekla Structures creates the part/bolt position dimensions. **None** creates no position dimensions. **Main part** creates the dimensions from the main part reference line. **Working points** creates the dimensions between the work points, such as the intersections of main and neighbor part reference lines.
Embedded objects	Creates position dimensions for locating embedded objects in cast unit drawings. The embedded objects are custom components attached to the cast unit. **As secondary objects** dimensions embedded objects in cast unit drawings the same way as secondary parts. **By reference point** dimensions embedded objects to their reference point, which is the origin of the custom component.
Secondary part	Creates dimensions to bolt holes or edges of the secondary part. **None** creates no position dimensions for secondary parts. **By bolt** dimensions bolt hole locations in the secondary parts. **By part** dimensions edges of secondary parts. **By both** dimensions bolt hole locations and edges of the secondary part.
Secondary part dimension direction	Aligns dimensions with the main or neighboring part. Only for skewed clip angles or shear plates.
Position from	Sets the start point for running dimensions. Only for skewed clip angles or shear plates bolted to a neighboring part.

Option	Description
Reversed direction for running dimensions	**Yes** changes the direction of running dimensions. With this option, you can set the zero point to the end of a member rather than to the start.
Main part bolt position	**On** creates dimensions to the bolt hole locations in the main part.
Main part skew position	**Yes** creates horizontal and vertical check dimensions representing the skew position of a brace. Created between the work points of the main part. Skew check dimensions are located in the front view. Their location depends on the **Main dimension position** setting selected on the **General** tab. When the main dimensions are above the part, the skew dimensions are below, and vice versa.
Skew position	Defines how Tekla Structures dimensions skewed secondary part positions. **None** creates no dimensions for skewed secondary part positions. **angle** creates an angle dimension for the secondary part. **Dimensions** creates dimensions for the skewed position of the secondary part. **Both** creates both the angle and the dimensions.
Centered part	Controls the dimensions of centrally-placed parts. These settings have an effect only if position dimensions are created. **Internal** dimensions the centered parts overall size. **Position** dimensions the part to the main part center lines. **None** creates no centered part dimensions.

Option	Description
Centered bolt	Controls the dimensions of centrally-placed bolt groups. **Internal** dimensions the centered bolts spread. **Position** dimensions the bolts to the main part center-lines. **Centered bolt** overrides the **Secondary part bolt internal** option for centered bolts. This only applies to bolts located centrally on the part.
Elevation dimensions	**On** creates elevation dimensions.
Combine equal dimensions	Combines equal dimensions. The choices are **Off**, **3*60** or **3*60=180**. The accuracy of combining equal dimensions is 0.1.
Minimum number to combine	Defines the minimum number of dimensions to combine.

See also

XS_PART_POSITION_TO_EDGE_NEAREST_TO_NEIGHBOUR (page 341)

XS_WORKING_POINTS_VALID_ALSO_OUTSIDE_PART (page 502)

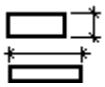
Dimensioning properties - Part dimensions tab (Integrated dimensioning)

Use the options on the **Part dimensions** tab of the **Dimensioning Properties** dialog box to view and modify part dimensioning settings for single-part, assembly and cast unit drawings.

Note that the contents of the dialog box vary depending on the drawing type, and all the options listed below are not available for all drawing types. This dialog box is displayed if you use **Integrated dimensions** dimensioning type.

Option	Description
Internal	Creates internal dimensions for secondary parts connected to the main part. **None** creates dimensions for secondary parts.

Option	Description
	Necessary creates only the dimensions that are necessary for assembling the parts. **All** creates all dimensions for the secondary parts.
Main part overall dimensions	**Once** creates one overall dimension for the main part. **All** creates overall dimensions for main parts in all views. **None** creates no overall dimensions for the main part. The **Assembly part overall dimensions** settings have some effect on these options.
Assembly part overall dimensions	**Length only** creates overall dimensions for an entire assembly or cast unit in the x direction only. **All dimensions** creates overall dimensions for an assembly or cast unit in all directions. **Off** creates no overall dimensions for an assembly or a cast unit.
Main part work points	**On** creates a check dimension between the outermost work points.
Main part shape (**Shape dimensions**)	**On** creates dimensions to show the shape of a main part. By default, Tekla Structures automatically draws shape dimensions on both ends of a beam, even if the ends are symmetrical.
Main part radius dimensions (**Radius dimensions**)	**On** creates radius dimensions for curved chamfers and round holes in the main part. This option is available only if you have set **Main part shape** dimensions to **On**. Note that this option does not create radius dimensions for curved beams or polybeams with chamfers.
Bevel dimensions	**On** creates linear dimensions of a bevel.

Option	Description
Bevel angle	Creates an angle dimension and defines which side of the bevel to dimension. The choices are **None**, **Angle of cut** and **Angle of beam**.
Knock off dimensions	**On** creates check dimensions from the edge of the main part to the work point.
Preferred dim side	Sets the preferred view (front or side) for part dimensions. 
From the nearest floor level to part	**On** creates dimensions indicating the distance from the closest floor level to the bottom and/or top of the parts.
From grid to part centerline	**On** creates dimensions showing the offset of a part from the grid to the part center line.
From grid to part ends	**On** creates dimensions showing the offset of a part from the grid to the near or far end of the part.

See also

XS_SINGLE_PART_SHAPE (page 428)

XS_EQUAL_SHAPE_DIMENSIONS_TO_BOTH_ENDS_LIMIT (page 250)

XS_CREATE_ROUND_HOLE_DIMENSIONS (page 137)

Dimensioning properties - Bolt dimensions tab (Integrated dimensioning)

Use the options on the **Bolt dimensions** tab of the **Dimensioning Properties** dialog box to select which bolt dimensions to create and how in single-part, assembly and cast unit drawings.

Note that the contents of the dialog box vary depending on the drawing type, and all the options listed below are not available for all drawing types. This dialog box is displayed if you use **Integrated dimensions** dimensioning type.

Option	Description
Main part bolt internal dimensions	Creates internal dimensions for bolt groups in the main part. **None** creates none of the internal bolt dimensions.

Option	Description
	Internal creates bolt group internal dimensions (distances between bolts). **All** creates edge distance and bolt group internal dimensions. Edge distance is the dimension from the outermost bolt to the part edge.
Main part bolt internal dimensions: **Skewed bolt group**	Indicates whether the dimensions are parallel to the part or the bolt group. The choices are **No dimensions**, **In part direction**, and **In bolt group direction**.
Secondary part bolt internal dimensions	Creates internal dimensions for bolt groups in the secondary part. The choices are **None**, **Necessary**, **Internal**, and **All**.
Secondary part bolt internal dimensions: **Skewed bolt group**	Aligns the bolt dimensions with the secondary part or bolt group. The choices are **In part direction**, **No dimensions**, and **In bolt group direction**
Distance between extreme bolts: **Extreme bolts**	Creates check dimension between the outermost bolts. The choices are **None**, **Main part**, and **Assembly**.
Distance between extreme bolts: **Extreme bolts to work points**	Creates check dimensions from the outermost bolts to the work points. **Yes** creates the check dimensions.
Preferred dim side	Sets the preferred view (front or side) for bolt dimensions.
Combine bolt dimensions	Sets the format of the combined bolt group internal dimensions. You can combine bolt group internal dimensions and display them in the format **3*60** or **3*60=180**, or have single dimensions.
Minimum number to combine	Defines the minimum number of dimensions to combine.

See also

XS_COMBINED_BOLT_DIM_CHARACTER (page 127)

Dimensioning properties - Dimension grouping tab (Integrated dimensioning)

Use the **Dimension grouping** tab in **Dimensioning Properties** dialog box to view and modify settings for dimension grouping in single-part, assembly and cast unit drawings.

Note that the contents of the dialog box vary depending on the drawing type, and all the options listed below are not available for all drawing types. This dialog box is displayed if you use **Integrated dimensions** dimensioning type in dimensioning.

Option	Description
Activate dimension grouping	Selects the objects for grouping.
Parts	Groups according to parts.
Bolts	Groups according to bolts.
Components	Groups according to components.
Cuts/Shapes	Groups according to cuts or shapes.
Automatic tagging	Defines how to display information in a dimension line.
Display tags	Displays tags.
Include part count in the tag	Includes part count in the tag.
Do not display marks\nfor the grouped items	Does not display part marks for grouped items.
Available elements	Elements available to define identical conditions.
Add >	Adds items to **Selected elements** list.
Remove	Removes items from **Selected elements** list.
Move up	Moves the element higher in the list.
Move down	Moves the element lower in the list.
Update grouping\nwhen model \nchanges	**Yes** updates dimension grouping automatically when the model changes.

Dimensioning properties - Sub-assemblies tab (Integrated dimensioning)

Use the options on the **Sub-assemblies** tab of the **Assembly - Dimensioning Properties** dialog box to view and modify which dimensions to create and how for sub-assemblies.

Note that the contents of the dialog box vary depending on the drawing type, and all the options listed below are not available for all drawing types. This dialog box is displayed if you use **Integrated dimensions** dimensioning type in dimensioning.

Option	Description
Dimension parts inside sub-assemblies	Defines whether to dimension parts inside sub-assemblies. **Yes** creates internal dimensions for parts inside sub-assemblies. **No** does not create internal dimensions for parts inside sub-assemblies.
Measure sub-assembly position from	Defines the position from which the sub-assembly is measured. **None** does not measure the sub-assembly position. **Bolt** measures the position of the sub-assembly from the bolts. If bolts are not included in the sub-assembly, or if it is not possible to measure the position from bolts, Tekla Structures measures the sub-assembly position from the reference point. **Extrema points** measures the position of the sub-assembly from the bounding box of the sub-assembly. **Reference point** measures the position of the sub-assembly from the reference point.

Dimensioning properties - Reinforcement dimensions tab (Integrated dimensioning)

Use the **Reinforcement dimensions** tab in the **Dimensioning Properties** dialog box to view and modify the setting affecting the creation of reinforcement dimensions and how they are displayed.

This dialog box with this tab is displayed if you use **Integrated dimensions** dimensioning type in dimensioning cast unit drawings.

Option	Description
Dimensions for reinforcing bar groups	**On** creates dimensions for reinforcing bar groups. This also activates the other selections on this tab.
Dimension mark settings	Sets the mark type, which can be dimension line, dimension mark, tagged dimension mark, or variants of these typical styles. The Default environment contains three predefined settings files: `rebar_dimension_line`, `rebar_dimension_mark`, and `rebar_tagged_dimension_mark` Clicking the **...** button opens the dimension properties, and you can view the settings file and also change the settings, if needed.

General arrangement drawing dimensioning properties

Use the **Grid** tab in the **General - Dimensioning Properties** dialog box to view and modify settings for grid and overall dimensions in general arrangement drawings.

To open this dialog box:

- IOn the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** and click **Dimensioning...**.
- In an open GA drawing, double-click the drawing background, and click **Dimensioning...**

Grid tab

Option	Description
Grid line dimensions	**On** creates grid dimension lines.
Overall dimension	**On** creates the overall dimensions.

Option	Description
Dimension positioning: **Horizontal**	Positions the vertical grid and overall dimension lines to the **Left** or **Right** side of the drawing or on **Both** sides.
Dimension positioning: **Vertical**	Positions the horizontal grid and overall dimension lines **Above** or **Below** the drawing or **Both**.

Parts tab

Option	Description
Maximum leader line length: **Outside dimensions**	Controls how close the dimension lines are positioned to the parts they are dimensioning. Defines that the outside dimension lines take the maximum leader line length from the grid line.
Maximum leader line length: **Inside dimensions**	Controls how close the dimension lines are positioned to the parts they are dimensioning. Defines that the inside dimension lines take the maximum leader line length from the part reference point.
Include parts not entirely in view	**On** dimensions the parts partly outside the view. **Off** will not dimension these parts.
Maximum number of outside dimensions	Specifies the maximum number of dimension lines allowed outside the grid. When you dimension different objects on different dimension lines, using this setting helps you to create cleaner drawings. Once the desired maximum is reached, Tekla Structures creates the dimensions inside the grid.
Object group dimensioning rules	Specifies different object groups to be dimensioned on different dimension lines.
Object group	The object group to be dimensioned.
Positioning	**No dimensions** does not create dimensions for the parts. **Inside grid** creates dimensions next to or near the parts being dimensioned. All part dimensions are placed inside the grid when the parts

Option	Description
	are inside the grid. The dimension will still be outside, even if you have selected **Inside grid** because of where the parts are in the end bay and the end being dimensioned to is next to the outside. **Outside grid** creates dimensions to parts and positions them outside the grid. **Either** creates dimensions to parts and positions them either inside or outside the grid depending on the part position and the setting of the **Maximum number of outside dimensions** option. You must use the **Either** option if you define the **Maximum number of outside dimensions**, so that Tekla Structures can position dimensions inside the grid when the maximum number of dimensions is reached outside the grid.
Horizontal position	**Left side** positions all dimensions to horizontal parts to the left of the grid. **Right side** positions all dimensions to horizontal parts to the right of the grid. **Distributed to both sides** positions all dimensions to horizontal parts to the grid nearest the part they are dimensioning.
Vertical position	**Above** positions all dimensions to vertical parts above the grid. **Below** positions all dimensions to vertical parts below the grid. **Distributed to both sides** positions all dimensions to vertical parts to the grid nearest the part they are dimensioning.

Dimensioning properties - Parts tab (GA drawings)

Use the **Parts** tab in the **General - Dimensioning Properties** dialog box to view and modify settings for part dimensions in general arrangement drawings.

To open this dialog box:

- On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** and click **Dimensioning...**
- In an open GA drawing, double-click the drawing background, and click **Dimensioning...**

Option	Description
Maximum leader line length: **Outside dimensions**	Controls how close the dimension lines are positioned to the parts they are dimensioning. Defines that the outside dimension lines take the maximum leader line length from the grid line.
Maximum leader line length: **Inside dimensions**	Controls how close the dimension lines are positioned to the parts they are dimensioning. Defines that the inside dimension lines take the maximum leader line length from the part reference point.
Include parts not entirely in view	**On** dimensions the parts partly outside the view. **Off** will not dimension these parts.
Maximum number of outside dimensions	Specifies the maximum number of dimension lines allowed outside the grid. When you dimension different objects on different dimension lines, using this setting helps you to create cleaner drawings. Once the desired maximum is reached, Tekla Structures creates the dimensions inside the grid.
Object group dimensioning rules	Specifies different object groups to be dimensioned on different dimension lines.
Object group	The object group to be dimensioned.
Positioning	**No dimensions** does not create dimensions for the parts. **Inside grid** creates dimensions next to or near the parts being dimensioned. All part dimensions are

Option	Description
	placed inside the grid when the parts are inside the grid. The dimension will still be outside, even if you have selected **Inside grid** because of where the parts are in the end bay and the end being dimensioned to is next to the outside. **Outside grid** creates dimensions to parts and positions them outside the grid. **Either** creates dimensions to parts and positions them either inside or outside the grid depending on the part position and the setting of the **Maximum number of outside dimensions** option. You must use the **Either** option if you define the **Maximum number of outside dimensions**, so that Tekla Structures can position dimensions inside the grid when the maximum number of dimensions is reached outside the grid.
Horizontal position	**Left side** positions all dimensions to horizontal parts to the left of the grid. **Right side** positions all dimensions to horizontal parts to the right of the grid. **Distributed to both sides** positions all dimensions to horizontal parts to the grid nearest the part they are dimensioning.
Vertical position	**Above** positions all dimensions to vertical parts above the grid. **Below** positions all dimensions to vertical parts below the grid. **Distributed to both sides** positions all dimensions to vertical parts to the grid nearest the part they are dimensioning.

Mark properties in drawings

You can adjust the mark appearance and contents before creating a drawing and also in an open drawing.

Click the links below to find out more:

- Mark properties - Content, General, Merging and Appearance tabs (page 734)
- Mark elements (page 740)
- Positioning properties of view label, section and detail marks (page 750)
- Drawing weld mark properties (page 750)
- Model weld mark visibility and appearance properties in drawings (page 753)
- Level mark properties (page 756)
- Leader line types (page 758)

Mark properties - Content, General, Merging and Appearance tabs

Use the **General**, **Merging** and **Content** tabs in the properties of different types of marks to view and modify the settings affecting the mark content and appearance. For some marks, there is a separate **Appearance** tab.

Note that all the listed settings are not available for all marks.

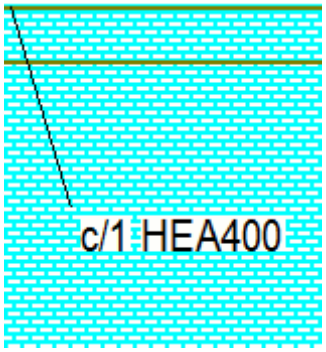
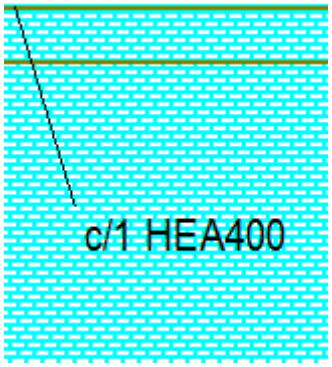
To open mark properties:

- On the **Drawings & reports** tab, click **Drawing properties**, select the drawing type, and go to mark properties directly by clicking the button of the respective mark (GA drawings) or through view properties.
- In an open drawing, double-click the drawing background, and go to mark properties directly by clicking the button of the respective mark (GA drawings) or through view properties.
- In an open drawing, double-click the drawing view frame and click the desired mark in the options tree on the left.
- In an open drawing, go to **Drawing** --> **Properties** and select a mark type.
- Double-click a mark in an open drawing.

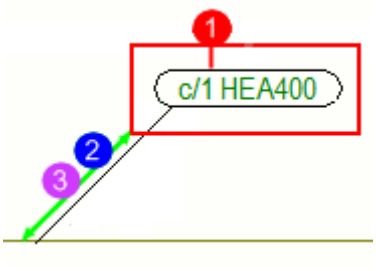
Setting	Description
Content tab:	

Setting	Description
Available elements **Elements in mark**	**Available elements** lists the elements that are available for the current mark. **Elements in mark** lists the elements that you have selected to include in the mark. For more information about mark elements, see Mark elements in drawing marks (page 740).
< Add frame	Add frames around individual elements in side the mark.
Frame around elements: **Type** and **Color**	Defines the element frame type and color for one or several elements. To select all elements in the **Elements in mark** list to apply the same modification to all of them, hold down **Shift** on the keyboard, and click the last element in the list.
Font: **Color**, **Height** and **Font**	Defines the font type, color, and height used in element texts in one or several elements. To select all elements in the **Elements in mark** list to apply the same modification to all of them, hold down **Shift** and click the last element in the list. Clicking **Select...** gives more font choices.
Units: **Units** and **Format**	Change the units and format of the length, height, spacing or diameter element that you have selected from the **Elements in mark** list.
General or **Appearance** tab:	
In main parts **In secondary parts** **In sub-assembly main parts** **In sub-assembly secondary parts**	These settings are for bolt marks. **Visible** displays bolt marks. **Not visible** does not display bolt marks.
Visibility in view	**distributed** distributes the marks in the view. Tekla Structures only creates marks that are not visible in the other views. **always** always creates marks in the view, irrespective the settings in other views. **preferred** acts as **distributed**, but the preferred view has a higher priority. Select **distributed** to one view only in a drawing. If you set other views to **distributed**, the marks are located only in the view that has the setting **Visibility in view** set to **preferred**. **none** creates no marks.

Setting	Description
Parts out of view plane	This setting is available only in view level properties. **Visible** displays marks outside of the view in the drawing. **Not visible** displays no marks outside the view.
Ignore size	This setting is for bolt marks. Filters standard-sized bolt marks out of drawings, which means that Tekla Structures does not display bolt marks of the defined size in drawings. There are some advanced options that can be used in addition to the **Ignore size** setting so that you first give the ignored size and then set the advanced option: XS_OMITTED_DIAMETER_TYPE (page 334) defines if the ignored size is the size of the bolt (value `BOLT`) or the hole (value `HOLE`). XS_OMITTED_BOLT_TYPE (page 334) can be used for filtering out ignored bolt marks on the basis of the bolt standard. Enter the name of the bolt standard as the value, for example, 7990. You can also use wildcards, such as * or ?. XS_OMITTED_BOLT_ASSEMBLY_TYPE (page 333) can be used for filtering out ignored bolt marks on the basis of the bolt type. The values are `SITE`, `SHOP`, and `SITE_AND_SHOP`.
Merge marks	This setting is for part marks and surface treatment marks. **On** merges marks. For more information about part marks merging distance and other merging principles, see . By default, the maximum merging distance is 1200 mm from the part.
Frame around mark: **Type** and **Color**	Define the frame to be used around marks, and the color of the frame.

Setting	Description
Background mask	Select **Opaque** to hide the part of the drawing that is covered by the mark.  Select **Transparent** to show the part of the drawing that is covered by the mark, so that line work is visible, for example. 
Leader line: **Rebar group mark**, **Merged marks**, **Type**, **Arrow** and **Use hidden lines for hidden parts**.	Define the leader line type and line arrow type, and select to hide leader lines for hidden parts. If you are merging marks, or adding rebar group marks, select one of the following options: : **One leader line to group** creates one leader line for a group of objects. : **One leader line per row** merges the marks and creates one leader line for a row of objects. : **Parallel leader lines** merges the marks and creates parallel leader lines. : **Leader lines to one point** merges the marks and draws all leader lines to one point.

Setting	Description
	: **Perpendicular leader lines** allows you to create marks with perpendicular leader lines to rebar groups. Creates marks with leader lines to each rebar in a group plane : **Leader line to first and last** allows you to create marks with perpendicular leader lines to rebar groups. Creates marks to first and last rebar in a group. You can control the length of the perpendicular leader lines with the advanced option XS_MARK_LEADER_LINE_LENGTH_FOR_PERPENDICULAR (page 306) (**File** --> **Settings** --> **Advanced options** --> **Marking: General**). The default value is 0 mm.
Rotation	Define the part mark rotation.
Alignment	Select one of the alignment options: **Left** **Center** **Right** **Leader line** The **Leader line** option is available in following mark types: bolt marks, connection marks, part marks, neighbor part marks, pour object marks, reinforcement marks, neighbor reinforcement marks, merged reinforcement marks and surface treatment marks.
Place...	Clicking the **Place...** button opens the placing dialog box. **Placing**: **free** allowsTekla Structures to search for the first suitable location for the mark. **Placing**: **fixed** allows you to place the mark in any location. When you use the option fixed, the mark stays where it is even though you update the drawing, whereas with free, Tekla Structures tries to find the optimal place for the annotation object. **Search margin** is the empty margin that you want to leave around the mark. **Minimum distance** is the minimum distance of the mark from the part.

Setting	Description
	Maximum distance is the maximum distance of the mark from the part. Note that if you use a high **Search margin** and **Minimum distance** values, the mark placement does not work properly. **Quarter** defines the areas where Tekla Structures searches for a position to place the mark.  **(1)** Search margin **(2)** Minimum distance **(3)** Maximum distance For more information about placing annotation objects, see Arrange annotation objects.

Setting	Image	Description
Merging tab Use the **Merging** tab in **Reinforcement mark** properties to view and modify the settings affecting the merging of the reinforcement marks in cast unit drawings. For more information about merged reinforcement marks, see Elements in merged reinforcement marks, and .		
Identical marks in same cast unit		**One leader line to group** creates one leader line for a group of reinforcing bars.
		One leader line per row merges the marks and creates one leader line for a row of reinforcing bars.
		Parallel leader lines merges the marks and creates parallel leader lines.

Setting	Image	Description
		Leader lines to one point merges the marks and draws all leader lines to one point.
		No merge does not merge marks. Tekla Structures creates an individual leader line for each mark. If you select **No merge**, you still need to define the mark content for the marks that Tekla Structures automatically merges on the **Merging** tab.
Preferred direction of merge		**Merge vertical** merges the marks in the vertical direction of the drawing.
		Merge horizontal merges the marks in the horizontal direction of the drawing.

Mark elements

Different kinds of marks have some elements in common and some of them are mark type specific. The common elements and the mark-specific elements are listed below.

Common elements in marks

There are some mark elements that can be used in most of the mark types.

Element	Description
Text	Adds a tex field inside the mark where you can add your text.
Symbol	Opens a dialog box where you can change the symbol file in use, and select a symbol from a Tekla Structures symbol file to add to the mark.
Template	Adds in the mark a custom graphical template created with **Template**

Element	Description
	editor. Opens a dialog box where you can select the template. For more information about adding templates in marks, see . In mark templates, you can include detailed information of an embed or assembly, such as the sub-material used. Or you can use a template that changes the unit and the number of decimals in measurement values in a mark. You can also add graphical objects using the **Template editor** tools.
Add space < >	Adds spaces between mark elements.
Remove space<--	Adds a backspace between the desired elements to remove the default space between them. The default space between the elements depends on the text height and can be changed with the advanced option XS_MARK_ELEMENT_SPACE_FACTOR (page 303).
Frame	Adds a frame around an element or the whole mark depending on your selection.
User-defined attributes **Template attributes**	Adds a user-defined attribute or template attribute in the mark. In marks, you cannot use template attributes such as MODEL_TOTAL that refer to the whole model. Marks only check the information from the object in the drawing and not from the whole model. For more information about adding user-defined attributes or template attributes in marks, see .
Custom properties	Adds a custom property in the mark.

Part mark elements
You can define part mark contents independently for main and secondary parts and for sub-assembly main and secondary parts.

The following table lists all elements specific to part marks and neighbor part marks.

Element	Description
Assembly position	Adds the prefix and position number of the assembly.
Part position	Adds the prefix and position number of the part.
Profile	Adds the profile name of part, assembly, or cast unit main part.
Material	Adds the material of part, assembly, or cast unit main part.
Name	Adds the name of part, assembly, or cast unit main part.
Class	Adds the class of part, assembly, or cast unit main part.
Finish	Adds the finish of part, assembly, or cast unit main part.
Size	Adds the size of the part or the assembly or cast unit main part.
Length	Adds the length of the part or assembly, or of the cast unit main part. You can change the unit and format of the length.
Camber	Adds the camber of the part or the assembly or cast unit main part (if this user-defined part attribute is set).
Fittings (NS/FS)	Displays the near side/far side marks in the part mark. (only available in front views).
Face direction	Displays main compass direction (North, East, South, West) of the face, where the mark is added. The direction can only be shown if • the face is vertical • the direction is the same for all assemblies with the same assembly position number In other cases, the element produces no text to mark. Furthermore, face direction is not shown for columns in general

Element	Description
	arrangement drawings, if you have set **Mark always to center of column in GA drawings** to **Yes** in **File menu** --> **Settings** --> **Options** --> **Orientation marks** .
Gage of outstanding leg	Adds the hole gage. You can control the format of this option with the advance option XS_GAGE_OF_OUTSTANDING_LEG_STRING (page 264).
Center-to-center distance	Adds center-to-center distance in the mark. You can control the format of this option with the advanced options XS_CENTER_TO_CENTER_DISTANCE_IN_ONE_PART_STRING (page 112) and XS_CENTER_TO_CENTER_DISTANCE_IN_TWO_PARTS_STRING (page 113).
Rotation angle	Adds the rotation angle of a spiral beam in the mark. For other parts this element gives an empty value.

Bolt mark elements

You can define bolt mark options independently for site and shop bolts.

Below is a list of the elements specific to bolt marks.

For additional information about defining size in bolt marks, see .

Element	Description
Bolt length	Adds the length of the bolt. You can change the unit and format of the length.
Bolt diameter	Adds the bolt diameter. You can change the unit and format of the diameter.
Hole diameter	Adds the hole diameter. You can change the unit and format of the diameter.
Hole depth	Adds the hole depth
Material	Adds the bolt material grade.
Standard	Adds the bolt standard.

Element	Description
Short name	Adds the bolt's short name. This can be the commercial name of a specific bolt, for example.
Full name	Adds the complete name of the bolt. This name is visible in the dialog box list.
Assembly type	Adds the bolt assembly type.
Number of bolts	Adds the quantity of bolts.
Slot length (x) **Slot length (y)**	Adds the slot length in the x or y direction. You can change the unit and format of the length.
Slot length	Adds the slot length. You can change the unit and format of the length.
Slot height	Adds the slot height. You can change the unit and format of the height.
Size	Adds the hole size. You can change the unit and format of the size.
Countersunk	Adds countersink in the countersunk bolt marks.
Gage of outstanding leg	Adds hole gage. You can control the format of this element with the advanced option XS_GAGE_OF_OUTSTANDING_LEG_STRING (page 264).
Center-to-center distance	Adds center-to-center distance. You can control the format of this element with the advanced options XS_CENTER_TO_CENTER_DISTANCE_IN_ONE_PART_STRING (page 112) and XS_CENTER_TO_CENTER_DISTANCE_IN_TWO_PARTS_STRING (page 113).

Reinforcement and neighbor reinforcement mark elements
You can define mark contents separately for single reinforcing bars, bar groups, and reinforcement meshes.

Below is a list of the elements you can include in all reinforcement and neighbor reinforcements marks.

Element	Description
Name	Adds the name of the bar or mesh.
Grade	Adds the material grade of the bar or mesh.
Layer	Adds the layer of the reinforcement.
Diameter	Adds the nominal diameter of the bar.
Class	Adds the class of the bar or mesh.
Length	Adds the total length of the bar. You can change the unit and format of the length.
Number	Adds the quantity of the bars.
Position	Adds the reinforcement position number.
Assembly position	Adds the position number of the associated rebar assembly.
Shape	Adds the shape of the bar or mesh.
Weight	Adds the weight of the bar or mesh.
cc	Adds the center-to-center spacing of the bars. The options are: • **cc** adds the spacing value if spacing does not vary • **cc min** adds the smallest spacing value of the bar group if spacing varies • **cc max** adds the largest spacing value of the bar group if spacing varies • **cc exact** lists all spacing values of the bar group • **cc target** lists all reinforcing bar target spacing values You can change the unit and format of the cc options.

Element	Description
Pullout picture	Adds a pull-out picture of a bar to the mark. For more information about pull-out pictures, see . You can also add pull-out pictures in a reinforcement mark in an open drawing.

Reinforcement and neighbor reinforcement mesh mark elements
You can define mark contents separately for reinforcement meshes.

The following elements are specific to the reinforcement and neighbor reinforcement mesh marks, other elements are the same as for reinforcement marks.

Element	Description
Size	Adds the nominal diameters of the mesh bars, dimensions of the mesh, and the spacings of the bars in the longitudinal and crossing directions.
Mesh length	Adds the length of the reinforcement mesh.
Mesh width	Adds the width of the reinforcement mesh.
cc	You can define cc individually for the longitudinal and crossing bars in the mesh. Adds the center-to-center spacing of the bars. The options are: • **cc longitudinal**/**cc crossing** adds the spacing value if spacing does not vary • **cc min longitudinal**/**cc min crossing** adds the smallest spacing value of the bar group if spacing varies • **cc max longitudinal**/**cc max crossing** adds the largest spacing value of the bar group if spacing varies • **cc exact longitudinal**/**cc exact crossing** lists all spacing values of the bar group

Element	Description
	• **cc target** lists all reinforcing bar target spacing values
Diameter longitudinal	Adds the diameter or size of the longitudinal bars.
Diameter cross	Adds the diameter or size of the crossing bars.

Elements in merged reinforcement marks
Some extra elements are available for merged reinforcement marks, in addition to basic reinforcement marks.

For more information about merging marks, see .

Element	Description
Block prefix	Adds text or a value to the beginning of each repeated block. Opens a dialog box where you can enter the prefix. You can use the following variables as block prefixes: `%NUMBER%` includes the number of marks merged to the mark. `%NUMBER_IN_PLANE%` includes the number of marks merged in the plane of the drawing to the mark. `%NUMBER_OUT_OF_PLANE%` includes the number of marks merged in the depth direction of the drawing to the mark.
Single mark content	Adds the content of the single reinforcement mark selected on the **Content** tab to the mark.
Distances between groups	Adds the center-to-center distances between reinforcing bars or bar groups included in a merged mark. Opens a dialog box where you can enter the values.
Symbol separating blocks in mark	Adds a symbol between the blocks in the merged mark. Opens a dialog box where you can define the symbol. The elements that appear before this element on the mark content list generate a block.

Connection mark elements
In connection marks, you can show the connection code, name, number, and running number, the group they belong to, potential errors, and the related DSTV code.

Below is a list of elements specific to connection marks.

Element	Description
Code	Adds the connection code. This is a user-defined code you give for the connection in the connection dialog box. The code can be either a text string or a number.
Name	Adds the connection name, for example, Tube_splice.
DSTV code	Adds the DSTV code.
Connection number	Adds the number of the connection.
Running number	Adds the connection running number. All connections are automatically numbered with a running number.
Group	Adds the connection group.
Connection error	Adds the connection error. The numbers correspond to the connection symbol colors: • 1 = green • 2 = yellow • 3 = red symbol

Pour object mark elements

Element	Description
Material	Adds the defined pour material.
Pour number	Adds the identifier that groups the pour objects in the same group, for example, to be poured at the same time.
Pour type	Adds a property of the pour based on the name of a part.
Concrete mixture	Adds the defined concrete mixture.

Surface treatment mark elements
In surface treatment marks, you can show the name, material, Tekla Structures-specific name, and the code of the surface treatment.

Element	Description
Name	Adds the name defined in the **Name** box in the surface treatment properties in a model.
Material	Adds the surface treatment material.
Class	Adds the class of the surface treatment.
Code	Adds the code of the surface treatment option selected from the **Subtype** list in the in the surface treatment properties in a model. For example, if the subtype is `MF Magnesium Float`, the code is `MF`.
Surface treatment name	Adds the full name of the surface treatment option selected from the **Subtype** list in the in the surface treatment properties in a model. For example, if the subtype is `MF Magnesium Float`, the full name is `Magnesium Float`.

Section and detail mark elements

In section and detail marks, you can show the name of the section/detail, the name of the current drawing, and the name of the source drawing.

Element	Description
Section name/Detail name	Adds the name of the section or the detail (A, B, C, and so on).
Drawing name	Adds the name of the current drawing.
Source drawing name	Adds the name of the drawing where the view is.
Source drawing name when moved	Adds the name of the drawing where the view is. This is shown only if the view is not in the same drawing as the section/detail mark.

View, section view and detail view label mark elements

In view labels, you can show the name of the view, section or detail, the view scale, drawing name, and source drawing name.

Element	Description
View name/Section name/Detail name	Adds the name of the view, section or the detail.
Scale	Adds the scale of the view.
Drawing name	Adds the name of the current drawing.
Source drawing name	Adds the name of the drawing where the view has originally been created.
Source drawing name when moved	Adds the name of the drawing where the view has originally been created. This is shown only when the view has been moved from its original drawing.

Positioning properties of view label, section and detail marks

Use the **Position** tab in the **Mark Contents** dialog box in view properties to set the positioning options for view labels marks, section marks and detail marks.

Option	Description
Show on	For section marks. Defines if the section marks are shown at both ends of the cut line or at the left or right end.
Text position	Defines the position of the mark text in relation to the line or in relation to the symbol or symbol center line. **Horizontal offset** sets the horizontal offset of the mark text from the line. **Vertical offset** sets the vertical offset of the mark text from the line.
Text rotation	For section marks. Defines the rotation of the mark text.
Alignment	For view label marks. Defines if the view label mark is aligned to the center, right, or left.

Drawing weld mark properties

In **Weld Mark Properties** dialog box you can view and modify the properties of a weld mark that has been manually added in a drawing.

To open the weld mark properties, do one of the following in an open drawing:

- Double-click a manually created weld.
- Hold down **Shift** and click **Weld mark** on the **Annotations** tab.
- On the **Drawing** tab, click **Properties** --> **Weld mark**.

Option	Description
Prefix	a= design throat thickness, s= penetration throat thickness, or z= leg length
Size	The size of the weld. If you select a partial penetration weld as the weld type, you can enter two sizes.
Type	The type of the weld. For a list of available weld types and their descriptions, see List of weld types. You can customize some of the weld type symbols, see Customize weld type symbols for more information.
Angle	The angle of weld preparation, bevels, or groove. Tekla Structures displays the angle between the weld type symbol and the fill type contour symbol.
Contour	The fill type contour of a weld can be: • None • Flush — • Convex ⌒ • Concave ⌣

Option	Description
Finish	Tekla Structures displays the finish symbol above the weld type symbol in drawings. The options are: • **G** (Grind) • **M** (Machine) • **C** (Chip) • (Flush finished weld) • (Smooth blended weld face)
Length	The length of a regular weld depends on the length of the connection between the welded parts. You can set the exact length of a polygon weld by, for example, defining the start and end points of the weld.
Pitch	The center-to-center spacing of welds for non-continuous welds. Pitch is shown in the weld mark if the value is greater than 0.0. To create a non-continuous weld, define the center-to-center spacing and the pitch of the welds. Tekla Structures calculates the distance between the welds as the pitch minus the length of the weld. By default, Tekla Structures uses the – character to separate weld length and pitch, for example, 50–100. To change the separator to @, for example, set the advanced option XS_WELD_LENGTH_CC_SEPARATOR_CHAR (page 501) to @.
Effective throat	The weld size used in weld strength calculation.
Root opening	The space between the welded parts.
Reference text	Additional information to appear in the weld symbol. For example, information about the weld specification or process.
Edge/Around	Indicates whether only one edge or the entire perimeter of a face should be welded. A circle in the weld symbol in drawings indicates the **Around** option has been used.
Workshop/Site	Indicates where the weld should be made.

Option	Description
Stitch weld	Set this option to **Yes** to create a staggered, intermittent weld. Stitch welds are staggered on both sides of the welded part. Tekla Structures shows the weld type symbols as staggered in weld symbols. If you set this option to **No**, a non- staggered intermittent weld is created. To show the pitch in a weld mark, set **Pitch** to a value greater than 0.0.
Placing	**Search margin** is the empty margin that you want to leave around the mark. **Minimum distance** is the minimum distance of the mark from the part. **Quarter** defines the areas Tekla Structures searches for a space to place the weld marks. **Placing** is the method used to place welds marks: • **free** lets Tekla Structures decide the location and direction of the mark. • **fixed** allows you to place the weld at any point.

Model weld mark properties in drawings

You can select which model weld marks are visible in drawings, and define the content that is displayed in weld marks. In assembly drawings, you can define the visibility of welds in sub-assemblies.

Use options in weld mark or welding mark properties to set the visibility and contents of model weld marks.

Single-part and assembly drawings weld mark properties

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
3. Click **Weld mark** in the options tree.

General arrangement drawing welding mark properties

1. Click **Drawings & reports** --> **Drawing properties** --> **GA drawing** .
2. Click **Weld mark**.

Object-level welding mark properties

- Double-click a model weld mark in a drawing, or go to **Quick Launch**, and start typing `welding mark` and select **Welding mark properties** from the displayed list.

The following table lists all model weld mark settings on all property levels (drawing, view, object). All levels do not contain all of the settings listed below.

Option	Description
Weld number	**Yes** shows the weld number. Tekla Structures assigns a number to each weld when the weld is created. You can select whether to show or hide the weld number.
Welds **Welds in sub-assemblies** (assembly drawings)	**None** displays no welds in the drawing.
	Site displays only site welds in the drawing.
	Workshop displays only workshop welds in the drawing.
	Both displays both site welds and workshop welds in the drawing.
Welds in hidden parts	Select how to show the weld marks for welds in hidden parts: • **None**: If the part is hidden, the weld mark is not drawn. • **Site**: If the part is hidden, weld marks for site welds are drawn only. • **Workshop**: If the part is hidden, weld marks for workshop welds are drawn only. • **Both**: Weld marks are always drawn for hidden parts.
Weld size limit	Enter a weld size to filter welds of that size or smaller out of the drawing. This is useful when you only want to show non-typical welds in a drawing. To set whether the weld size limit is an exact or minimum value, use the advanced option XS_WELD_FILTER_TYPE (page 500). To filter out a standard weld types, use the advanced option XS_OMITTED_WELD_TYPES (page 335).

Option	Description
Above line, **Below line** and **Other**	If there is no check mark in the **Visible** column next to the following properties, the properties are not displayed in the weld mark. These settings are set separately to above line and below line: **Prefix** **Size** **Type** **Angle** **Contour** **Finish** **Length** **Pitch** **Effective throat** **Root opening**
Reference text **Edge/Around** **Workshop/Site**	These settings are common to above line and below line. If there is no check mark in the **Visible** column next these properties, they are not displayed in the weld mark. The maximum number of characters that can be shown for **Reference text** is 80, including one character for each line of text. This limitation comes from the model weld.
Place...	**Placing**: • **free** allows Tekla Structures to search for the first suitable location for the mark, dimension, weld or another annotation object. **fixed** allows you to place the mark, dimension, weld or another annotation object in any location. • When you use the option **fixed**, the annotation object stays where it is even though you update the drawing, whereas with **free**, Tekla Structures tries to find the optimal place for the annotation object.

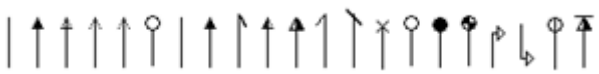
Option	Description
	Search margin is the empty margin that you want to leave around the mark. **Quarter** defines the areas Tekla Structures searches for a space for placing the mark. **Minimum distance** is the minimum distance of the mark from the part. Note that if you use high **Search margin** and **Minimum distance** values, the mark placement may not work properly.
Color	Sets the color of the text.
Height	Sets the height of the text.
Font	Sets the font of the text. Click **Select...** to show more options.
Type	Sets the type of the line.
Color	Sets the color of the line.
Background mask	Select **Opaque** to hide the part of the drawing that is covered by the mark. Select **Transparent** to show the part of the drawing that is covered by the mark, so that line work is visible, for example.

Level mark properties in drawings

Use the settings in the **Level Mark Properties** dialog box to view and modify level mark contents and appearance.

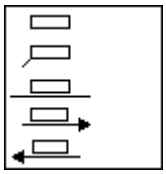
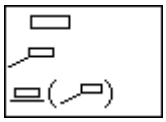
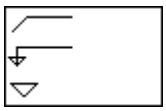
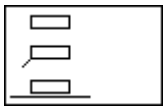
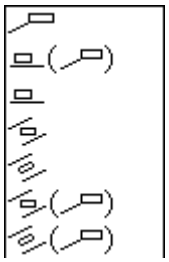
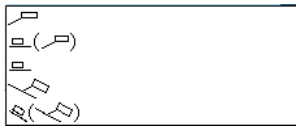
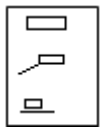
To open the dialog box in an open drawing, go to the **Drawing** tab, and click **Properties** --> **Level mark** .

Option	Description
General tab	
Prefix	Displays text before the mark.
Prefix for positive level	**+** displays a + character in front of the value.
Visibility of numeric value	Defines if the numeric values are visible or hidden.
Postfix	Displays text after the mark.

Option	Description
Level mark format: **Precision**	Defines the precision of the level mark dimension.
Level mark format: **Format**	Defines the format of the level mark dimension.
Level mark format: **Use grouping**	Defines whether to use different grouping options to represent the level mark dimensions.
Level mark format: **Units**	Defines the units used in level mark dimensions. The available values are **automatic**, **mm**, **cm**, **m**, **foot - inch**, **inch** and **feet**.
Placing	**Search margin** is the farthest distance Tekla Structures uses when searching for an empty space for the level mark. **Minimum distance** is the closest distance Tekla Structures uses to search for an empty space for a level mark. **Quarter** defines the areas Tekla Structures searches for a space to place the level marks. **Placing** is the method used to place level marks: • **free** lets Tekla Structures decide the location of the level mark. • **fixed** allows you to place the level mark at any point.
Appearance tab	
Text: **Color**, **Height**, **Font**, **Angle**	Defines the text color, height, font and angle.
Frame: **Type**, **Leader line**, **Color**	Defines the frame to be used around marks, the type of the leader line, and the color of the frame.
Frame: **Background mask**	Select **Opaque** to hide the part of the drawing that is covered by the level mark. Select **Transparent** to show the part of the drawing that is covered by the mark.
Arrow: **Type**	Defines the type of the arrow. 
	Defines the height and the length of the arrowhead.

Leader line types

You can use leader lines with text, symbols, associative notes and marks to make it clearer to which item it is related to.

Option	Description	Click the links below to find out more
Texts		
Symbols		
Level marks		
Part marks		
Revision marks		
Surface treatment marks		
Reinforcement marks		
Associative notes		

Part and neighbor part properties in drawings

Use the options in part properties or neighbor part properties to check and change the part or neighbor part properties. In neighbor part properties, you can also control the visibility and appearance of neighbor part bolts.

To go to part or neighbor part properties:

- On the **Drawings & reports** tab, click **Drawing properties** , select the drawing type and navigate to part or neighbor part properties.
- In an open drawing, double-click the drawing view frame, and click **Part** or **Neighbor part** in the options tree.
- In an open drawing, double-click the drawing background, and navigate to part or neighbor part properties.
- Double-click a part or neighbor part in an open drawing.
- In **Quick Launch**, start typing `part properties`, and then select **Drawing part properties** from the list.

The following table lists all settings on all property levels (drawing, view, object). All part properties do not contain all of the settings listed below.

Content tab:

Option	Description
Representation	**Outline** shows parts as solid objects. **Exact** shows parts as solid objects. This choice also draws the fillet edges and chamfers in the profile cross sections. For some profiles, **Outline** shows them as well. **Symbol** draws parts as lines. **Symbol with partial profile** shows a partial profile of the part. You can also adjust the length of the partial profile (**Length**), and the offset of the partial profile from the middle point of the part (**Offset from middle point**). **Workshop form** draws round tube profiles as wrap-around templates. Note that **Workshop form** can only be used in single part drawings. **Bounding box** draws parts as boxes surrounding the actual profiles.

Option	Description
	Bounding box is a good option to use with complex items with an extrema box containing many polygons slowing down drawings, because **Bounding box** makes drawings faster. **Base box** shows parts as boxes and uses the **h** and **b** values from the profile catalog as box dimensions.
Symbol offset	Defines the distance of the end points of reference lines and center lines from the end points of the object. Note that the **Symbol offset** value affects the size of the center line cross. If the value is 0, the cross is not visible.
Inner contours	Shows the inner contours of a tube.
Hidden lines	If the **Hidden lines** check box is selected, Tekla Structures shows hidden lines in secondary and neighbor parts. If the **Own hidden lines** check box is selected, Tekla Structures shows hidden lines in main parts.
Center line	Select whether you want to show center lines. Select the **Main part**: **Beam**, **Plate** or **Polygon** check box to show center lines in main parts. Select the **Sec part**: **Beam**, **Plate** or **Polygon** check box to show center lines in secondary parts. Note that the center line is only shown for assembly primary parts and not for secondary parts when looking from the cross section direction. If the part is viewed from the side, the center line is also shown for secondary parts. Note that the **Symbol offset** value affects the size of the center line cross. If the value is 0, the cross is not visible.

Option	Description
Reference lines	Select wether you want to show reference lines. Select the **Main part**: **Beam**, **Plate** or **Polygon** check box to show reference lines in main parts. Select the **Sec part**: **Beam**, **Plate** or **Polygon** check box to show reference lines in secondary parts.
Additional marks	Select the following check boxes to show additional marks in drawings: • **Orientation marks** shows orientation marks. • **Connecting side marks** shows connecting side marks. • **Pop-marks** shows pop-marks defined in NC settings. • **Edge chamfers** shows edge chamfers. • **Fillet edges** shows fillet edges.
Bolt representation (neighbor parts)	Select bolt representation. The options are **solid**, **exact solid Symbol**, **Symbol2** , **Symbol3**, **DIN symbol**, and **User-defined symbol**. **DIN symbol** corresponds to German standards (DIN). The only DIN symbols you can control are: • Symbol 24 for normal workshop bolts • Symbol 25 for normal site bolts • Symbol 26 for front countersunk site bolts • Symbol 27 for back countersunk site bolts • Symbol 28 for front countersunk workshop bolts • Symbol 29 for back countersunk workshop bolts • Symbol 30 for front countersunk holes

Option	Description
	• Symbol 31 for back countersunk holes **User-defined symbol** is a symbol that has been created in Symbol Editor.
Symbol content (neighbor parts)	Select whether to include **hole** or **axis** in the symbol.

Neighbor parts have a **Visibility** tab:

Option	Description
Neighbor parts	**None** does not show neighbor parts. **Connected parts** shows all parts connected to the model object. **Connecting parts** shows only the parts the model object is connected to. **All components**combines the **Connected parts** and the **Connecting parts** options. **By extreme** shows all parts within the boundaries of the main and secondary part.
Main/Secondary parts	**Main parts** shows only neighbor parts that make the main part of an assembly or a cast unit. **Secondary parts** shows only neighbor parts that are secondary parts of an assembly or a cast unit. **Both** shows both main and secondary parts.
Skew parts	**Yes** shows skew parts as neighbor parts in the drawing, **No** does not.
Bolts	**Yes** shows the bolts in the neighbor parts, **No** does not.

The **Appearance** tab is similar in all of the properties of all kinds of building objects (parts, neighbor parts, bolts, welds, surface treatments, reinforcement, and meshes).

Option	Description
Visible lines	Sets the **Color** and **Type** of the visible lines.

Option	Description
Hidden lines, Center line	Sets the **Color** and **Type** of the hidden lines. Sets the **Color** of the center lines.
Reference lines	Sets the **Color** and **Type** of the reference lines.
Text: **Color**	Sets the **Color** of the text.
Text: **Height**	Sets the **Height** of the text.
Text: **Font**	Sets the **Font** of the text. Click **Select...** to show more options.
Line: **Type**	Sets the **Type** of the line.
Line: **Color**	Sets the **Color** of the line.
Bolts: **Color**	Sets the color of the bolts in neighbor parts.

Both parts and neighbor parts have a **Fill** tab. To add a fill for outer part faces, use the **Part faces** area, and to add a fill to cross sections in section views, use the **Sections** area.

Setting	Description
Type	Defines the fill type. Clicking the button next to the list opens a preview of the hatch patterns. **Automatic** selects the fill type automatically from the hatch pattern schema files. **None** uses no fill.
Color	Defines the color for the fill. You can select a predefined color or use **Special** color that is not converted to black in printouts.
Background	Defines the background color for the fill. Background color selection is disabled for hardware hatches. Background color can be set for automatic hatches, but it has effect only if automatic hatch is not defined for the material in the hatch pattern schema file.

Setting	Description
Scale	**Automatic** scales and rotates the fill automatically. **Custom** allows you to select scaling and rotation manually. **Scaling in direction x** and **Scaling in direction y** define the scales in x and y direction. **Keep ratio of x and y** retains the relative proportions in the hatch pattern. **Angle** rotates the fill. Angle 0.0 is for horizontal and 90.0 for vertical.

Bolt properties in drawings

Use the options in bolt properties to check or change the bolt content and appearance.

To go to bolt properties:

- On the **Drawings & reports** tab, click **Drawing properties** , select the drawing type and navigate to bolt properties.
- In an open drawing, double-click the drawing view frame and select **Bolt** in the options tree.
- In an open drawing, double-click the drawing background, and navigate to bolt properties.
- Double-click a bolt in an open drawing.
- In an open drawing, go to **Quick Launch**, start typing `bolt properties`, and select **Drawing bolt properties** in the list.

The following table list all bolt settings on all property levels (drawing, view, object). All bolt properties do not contain all of the settings listed below.

Setting	Description
Solid/Symbol or **Bolt representation**	The options are **solid**, **exact solid**, **Symbol**, **Symbol2**, **Symbol3**, **DIN symbol**, and **User-defined symbol**. **DIN symbol** corresponds to German standards (DIN). The only DIN symbols you can control are in the following symbols in the `xsteel.sym` file:

Setting	Description
	• Symbol 24 for normal workshop bolts • Symbol 25 for normal site bolts • Symbol 26 for front countersunk site bolts • Symbol 27 for back countersunk site bolts • Symbol 28 for front countersunk workshop bolts • Symbol 29 for back countersunk workshop bolts • Symbol 30 for front countersunk holes • Symbol 31 for back countersunk holes **User-defined symbol** is a symbol that has been created in Symbol Editor.
Symbol content	Indicates whether to include the **Hole** and **Axis** symbols in the drawing.
Visibility of bolts	These settings are not available in object level bolt properties. Control the visibility of bolts **In main parts**, **In secondary parts** and **In sub-assemblies** separately. **Visible** shows bolt group holes in main parts or secondary parts. **Not visible** hides them. In assembly drawings you can also define whether to show or hide bolt group holes in sub-assemblies.
Color	Change the bolt color.

Surface treatment properties in drawings

Use the settings in surface treatment properties to check or change the surface treatment drawing properties.

To go to surface treatment properties:

- On the **Drawings & reports** tab, click **Drawing properties** , select the drawing type, and navigate to surface treatment properties.

- In an open drawing, double-click the drawing view frame and select **Surface treatment** in the options tree.
- In an open drawing, double-click the drawing background, and navigate to surface treatment properties.
- Double-click surface treatment in an open drawing.
- In an open drawing, go to **Quick Launch**, start typing `surface treatment`, and select **Drawing surface treatment properties** in the list.

The following table lists all surface treatment properties on all property levels (drawing, view, object).

Setting	Description
Visibility	**Visible** shows the surface treatment in the drawing. **Not visible** does not show the surface treatment in the drawing.
Representation	Defines the representation style of the surface treatment. The available options are **Outline**, **Exact**, **Workshop form**, **Symbol**, **Bounding box**, and **Base box**.
Show pattern	Defines whether the hatch pattern is shown.
Hidden lines	Defines whether hidden lines in secondary and neighboring parts are shown.
Own hidden lines	Defines whether hidden lines in main parts are shown.

Surface treatment hatch pattern properties (surfacing.htc)

You can change the properties of the hatch patterns for each surface treatment type separately.

The properties of the hatch patterns are defined in the `surfacing.htc` file, located by default in `..\Tekla Structures\<version>\environments\common\system`. In addition to this file, the surface treatment code file `product_finishes.dat` is needed. It is located in the same folder.

If you create your own surface treatment hatch patterns in your company, you can store `surfacing.htc` and `product_finishes.dat` in the firm folder defined by the advanced option `XS_FIRM`.

Note:

NOTE When you edit a schema file, you must reopen the model to apply the changes.

The syntax of `surfacing.htc` file is:

```
Surfacing Type, Surfacing Code, Hatch name, Scale, [Color],
[Automatic Scaling and Rotation]
```

Example:

```
1,MF,ANSI31,0.7
1,SMF,ANSI32,0.7
1,WT,ANSI33,0.7
1,HT,ANSI34,0.7
1,LSB,AR-SAND,0.7
2,SM1,CROSS,1.0
2,SM2,CHECKERED,1.0
3,TS3,FBBRICKC,1.0
4,FP,ANSI31,1.0
4,UP,ANSI32,1.0
```

Option	Description
`Surface treatment type`	• 1 = concrete finish • 2 = special mix • 3 = tile surface • 4 = steel finish
`Surface treatment code`	This is the abbreviation used in drawings and reports, for example, MF for Magnesium Float. The `product_finishes.dat` file contains a full list of surface treatment codes.
`Hatch name`	You can check the hatch pattern names and related hatch patterns by going to the **Fill** tab in part properties, selecting a hatch pattern from the **Type** list and clicking the **...** button next to the list. The selected hatch pattern is marked with a red frame.
`Scale`	Scale is a numeric value that Tekla Structures uses to scale hatches.
`Color` (optional)	0=black (default) 1=white

Option	Description
	2=red 3=green 4=blue 5=cyan 6=yellow 7=magenta 120= Special (use this color for gray shade) The hatch color defines the line width for the printer. If you do not define a color for a hatch in the `surfacing.htc` file, Tekla Structures uses the color defined on the **Appearance** tab in surface treatment properties. The **Visible lines** color and type is used for the front of the surface treatment, and the **Hidden lines** for the back.
`Automatic Scaling and Rotation` (optional)	1=true 0=false (default)

Reinforcement object properties in drawings

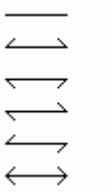
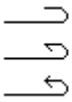
Use the options in the **Reinforcement** or **Neighbor Reinforcement** properties to check and modify the reinforcement, mesh, and rebar assembly visibility, appearance and content in drawings.

To go to reinforcement properties:

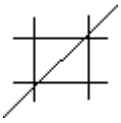
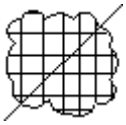
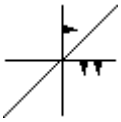
- On the **Drawings & reports** tab, click **Drawing properties** , select the drawing type, and navigate to reinforcement properties.
- In an open drawing, double-click the drawing view frame and select **Reinforcement** in the options tree.
- In an open drawing, double-click the drawing background, and navigate to reinforcement properties.
- Double-click reinforcement in an open drawing.
- In an open drawing, go to **Quick Launch**, start typing `reinforcement properties`, and select **Drawing reinforcement properties** in the list. If you want to open mesh properties instead, start typing `mesh properties`, and select **Drawing reinforcement mesh properties** instead.

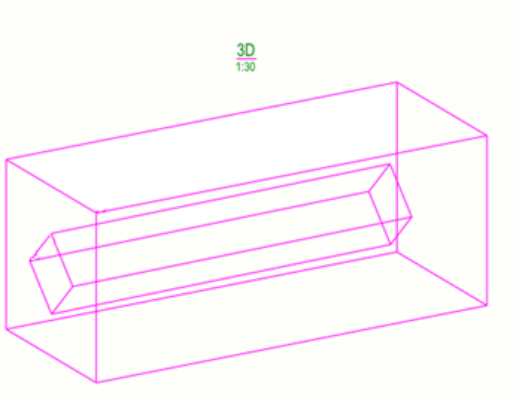
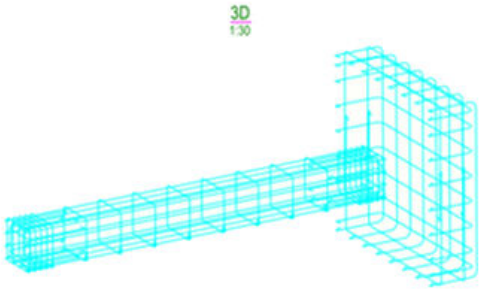
The following table lists all reinforcement properties on all property levels (drawing, view, object).

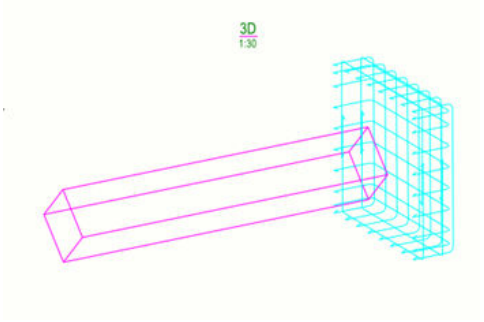
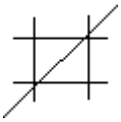
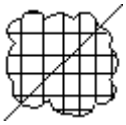
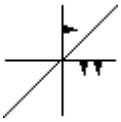
Option	Description
Bar content or **Content** tab:	
Visibility of all reinforcing bars	These settings are not available on object level. **Visible** shows the bars or meshes. **Not visible** does not show the bars or meshes
Reinforcing bar representation or **Representation**	**single line** draws a single line with radiused bends. **single line with filled ends** draws a single line for parallel bars and filled ends for perpendicular bars. **double lines** draws an outline of the bar with radiused bends. **double lines with filled ends** draws an outline of the bar with radiused bends and filled bar ends. **filled line** draws a solid bar with radiused bends. **stick** draws a single line without radiused bends. **outline** shows the shape of the mesh using an outline rectangle or polygon, and a diagonal line. Applies only to reinforcement meshes. **outline (ignore holes)** ignores holes and draws over them. Applies only to reinforcement meshes.
Visibility of reinforcing bars in group	**all** shows all bars in a group or mesh. **first bar** shows only the first bar in the group or mesh. **last bar** shows only the last bar in the group or mesh. **first and last** shows the first and last bar in the group or mesh. **bar in the middle of group** shows one bar in the middle of the group or mesh.

Option	Description
	two bars in the middle of group shows two bars in the middle of the group or mesh. **customized** indicates that you have specified the location of the only visible reinforcing bar. Applies only to bar groups and meshes.
Hide lines behind parts	Hides the lines behind the part. This is useful, for example, with lifting anchors where the reinforcing bar is partly outside the part. If you wish to show embeds while hiding reinforcement, use the XS_HIDDEN_LINES_UNHIDE_EMBEDDED (page 271) advanced option.
Hide lines behind other rebars	Hides the lines behind other reinforcement bar lines.
Symbol at straight end	 Reinforcing bar end symbols are always drawn in solid line type, regardless of the selected reinforcement line type. Applies only to reinforcing bars.
Symbol at hooked end	 Applies only to reinforcing bars. Reinforcing bar end symbols are always drawn in solid line type, regardless of the selected reinforcement line type.
Mesh content or **Content** tab:	
Visibility of all meshes	These settings are not available on object level.

Option	Description
	Visible shows the meshes in the drawing. **Not visible** does not show the meshes in the drawing.
Representation	**single line** draws a single line with radiused bends. **single line with filled ends** draws a single line for parallel bars and filled ends for perpendicular bars. **double lines** draws an outline of the bar with radiused bends. **double lines with filled ends** draws an outline of the bar with radiused bends and filled bar ends. **filled line** draws a solid bar with radiused bends. **stick** draws a single line without radiused bends. **outline** shows the shape of the mesh using an outline rectangle or polygon, and a diagonal line. Applies only to reinforcement meshes. **outline (ignore holes)** ignores holes and draws over them. Applies only to reinforcement meshes.
Visibility of longitudinal wires **Visibility of crossing bars**	**all** shows all bars in a group or mesh. **first bar** shows only the first bar in the group or mesh. **last bar** shows only the last bar in the group or mesh. **first and last** shows the first and last bar in the group or mesh. **bar in the middle of group** shows one bar in the middle of the group or mesh. **two bars in the middle of group** shows two bars in the middle of the group or mesh. **customized** indicates that you have specified the location of the only

Option	Description
	visible reinforcing bar. Applies only to bar groups and meshes.
Hide lines behind parts	Hides the lines behind the part. This is useful, for example, with lifting anchors where the reinforcing bar is partly outside the part. If you wish to show embeds while hiding reinforcement, use the XS_HIDDEN_LINES_UNHIDE_EMBEDDED (page 271) advanced option.
Hide lines behind other rebars	Hides the lines behind other reinforcement bar lines.
Mesh symbol	Defines the mesh symbol to be used. The mesh symbol appears in the middle of the diagonal line. **Symbol 1**  **Symbol 2**  **Symbol 3** 
Mesh symbol size	Defines the size of the mesh symbol.
Bar appearance, **Mesh appearance** or **Appearance** tab:	
Visible lines	Defines the color and type of the visible lines. Reinforcing bar end symbols are always drawn in solid line type, regardless of the selected reinforcement line type.
Hidden lines	Defines the color and type of the hidden lines.
Rebar assembly tab:	

Option	Description
Visibility of rebar assemblies	**Visible** shows the rebar assemblies. **Not visible** does not show the rebar assemblies.
Rebar **Representation**	**Outline**: All sub-assemblies of the rebar assembly are shown as outline boxes.  **As individual rebar objects**: All rebar objects in the rebar assembly and sub-assemblies are visible.  • **Sub-assemblies as outline, highest level as individual rebar objects**: All sub-assemblies are shown as outline boxes and the highest level of the rebar

Option	Description
	assembly is shown as individual rebar objects. 
Visible lines	Defines the color and type of the visible lines in rebar assemblies. Reinforcing bar end symbols are always drawn in solid line type, regardless of the selected reinforcement line type.
Mesh symbol	Defines the mesh symbol to be used in rebar assemblies. The mesh symbol appears in the middle of the diagonal line. **Symbol 1**  **Symbol 2**  **Symbol 3** 
Mesh symbol size	Defines the size of the mesh symbol in rebar assemblies.

Additional ways of modifying reinforcement

In addition to the settings in reinforcement properties, you can modify the reinforcement in any of the following ways:

- Increase the size of the bend and end symbols (in drawing units) using the advanced options XS_REBAR_BEND_MARK_SYMBOL_MIN_SIZE (page 362)

and XS_REBAR_END_SYMBOL_MIN_SIZE (page 364) in **File menu** --> **Settings** --> **Advanced options** --> **Concrete Detailing** .

- Use the advanced option XS_REBAR_REVERSE_END_SYMBOLS (page 367) in **File menu** --> **Settings** --> **Advanced options** --> **Concrete Detailing** to change the end symbols to a different direction.
- Modify the reinforcing bar bending schedule, rounding of bar dimensions, symbols for meshes, strands, and unbonding, and appearance of reinforcement pull-outs in the `rebar_config.inp (page 775)` file.

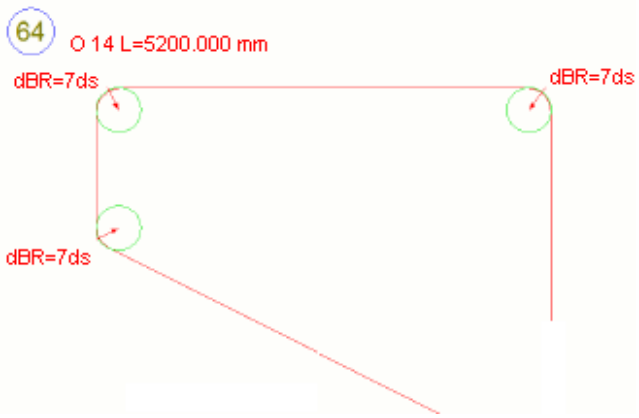
Reinforcement settings for drawings (rebar_config.inp)

Tekla Structures uses the settings in the `rebar_config.inp` file in the system folder (XS_SYSTEM (page 446)), firm folder or in the project folder to define the following reinforcement-related issues in drawings:

- Selected area-specific reinforcing bar bending schedule
- Rounding of bar dimensions
- Available symbols for meshes, strands, and unbonding
- Appearance of reinforcement pull-outs

The entries in the `rebar_config.inp` are listed and described below:

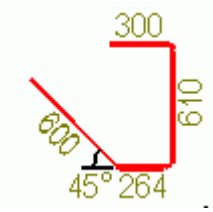
Entry	Description
`MergeOneFormat`	No longer used. Define these properties in the drawing properties.
`MergeTwoOrMoreFormats`	
`MergeAndFormat`	
`LeaderLinetype`	
`DimensionMarkSpacingSeparator`	`="/"` Affects the separator in reinforcement marks.
`ExactDimensionMarkSpacingSeparator`	`=" + "` Separator between different exact spacing values in reinforcement marks.
`ExactDimensionMarkPcsSeparator`	`="*"` Separator between the number of bars and their exact spacing value in reinforcement mark.
`BendingAngleTolerance`	Set a tolerance value for the angle. Angles that differ from the set tolerance value less than the tolerance are recognized and lead to a correct bending shape. Enter the tolerance value as a radians, not as a degree. The default value is 0.001 radians,

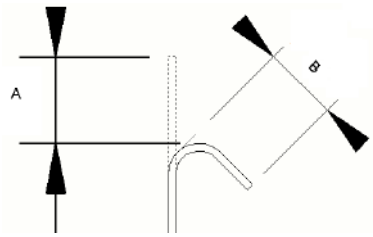
Entry	Description
	which is 0.0573 in degrees. This applies to all bending shapes.
BentRebarTolerance	Set a tolerance value. Depending on the value, slightly curved reinforcing bars get straight shape. If reinforcing bar diameter is 20 mm and radius is 200 m then value 20/200000 = 0.0001. This variable defines correct curved reinforcing bar in case of long reinforcing bars to get a correct shape for the bar. This option is used in comparing the relation of the reinforcing bar diameter and radius. If the relation is smaller than BentRebarTolerance, the reinforcing bar is of bend_type_1, otherwise it is of bend_type_34.
PullOutBendingRadiusAsMultiplier	Set to 1 to show the pullout bending radii using multiplier instead of mm. 
GroupBarMark	No longer in use.
MarkingDimAttributes	No longer in use.
ScheduleCountry	Defines which bending schedule is used. Affects bending shapes in templates and reports. The available schedules are FIN, SWE, UK, US. When you number the model, the bending shape for the bar is given according to this information. For example, in the Default environment, the bending shapes are letters A, B, C, and so on.

Entry	Description
ScheduleDimensionRoundingDirection ScheduleTotalLengthRoundingDirection	Options: • "UP": rounds bar dimensions up • "DOWN": rounds bar dimensions down • "NEAREST": rounds bar dimensions either up or down
ScheduleDimensionRoundingAccuracy	Sets the rounding accuracy for bar dimensions. Default is 1 mm. Tekla Structures rounds individual bar dimensions up or down according to the option you select for ScheduleDimensionRoundingDirection.
ScheduleTotalLengthRoundingAccuracy	Sets the rounding accuracy for the total bar length. Default is 10 mm. Tekla Structures rounds individual bar dimensions up or down according to the option you select for ScheduleTotalLengthRoundingDirection.
BentSymbolFile	Points to the symbol file that contains the available reinforcing bar bent symbols. By default, points to the file bent.sym, which in the default environment is located in the folder ..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\symbols.
MeshSymbolFile	Points to the mesh symbol file that contains the available mesh symbols. Affects the available reinforcement mesh symbols in drawings. By default, points to the mesh.sym file in the ...\Tekla Structures\<version>\environments\common\symbols folder.
StrandSymbolFile	Points to the strand symbol file that contains the available strand symbols. Affects drawings. By default, points to the strand.sym file in the ...\Tekla Structures\<version>\environments\common\symbols folder.
UnbondingSymbolFile	Points to the unbonding symbol file that contains the available unbonding symbols.

Entry	Description
RebarMeshSize	Template for rebar mesh size. ="%CC_DIAMETER_LONG%/%CC_DIAMETER_CROSS%-%CC_PITCHING_LONG%/%CC_PITCHING_CROSS%-%LENGTH%/%WIDTH%"
PullOutDimensionFormat	Defines the format for displaying the dimensions. The format follows the dimension properties format. Options: • 0 = ### • 1 = ###[.#] • 2 = ###.# • 3 = ###[.##] • 4 = ###.## • 5 = ###[.###] • 6 = ###.### • 7 = ### #/# • 8 = ###/##.###
PullOutDimensionPrecision	Sets the level of precision. The precision is calculated using the following formula: 1/value = precision. In metric systems, you may want to use the values 1, 10, and 100, and in imperial systems, the values 2, 4, 8, 16 and 32, for example.
PullOutDimensionUnit	Defines the units to use. Options: • 0 = auto • 1 = mm • 2 = cm • 3 = m • 4 = inch • 5 = foot and inch

Entry	Description
PullOutColor	Sets the color for the pull-outs in reinforcement marks. Options: • 1 = black • 2 = red • 3 = bright green • 4 = blue • 5 = cyan • 6 = yellow • 7 = magenta
PullOutVisibleLineType	Sets the line type for reinforcing bar shape in pull-outs. Options: 1 = 2 = 3 = 4 = 5 = 6 = 7 =
PullOutRepresentation	Sets the representation type. Options: • 0 = single • 1 = double • 2 = filled • 3 = stick
PullOutAngleColor	Sets the color for the angle in pull-outs. Options: • 1 = black • 2 = red

Entry	Description
	• 3 = green • 4 = blue • 5 = cyan • 6 = yellow • 7 = magenta • 8 = brown • 9 = green • 10 = dark blue • 11 = forest green • 12 = orange • 13 = gray 
PullOutAngleLineType	Sets the line type for angle lines in pull-outs. Options: • 1 = —————— • 2 = - - - - - • 3 = — — — • 4 = — - — - • 5 = - - - - - - - - • 6 = — ·· — ·· • 7 = — - - - -
PullOutLeaderLineMinLength	Sets a minimum length for the small leader lines that point to the dimension text. The default value is 10 mm. To switch leader lines off completely, use a large value.
PullOutShowDuplicateDims	Defines whether duplicate dimensions are showed multiple times for one bar. Options:

Entry	Description
	• 0 = duplicate dimensions are not shown (default) • 1 = equal and parallel dimensions are shown, but similar hook dimensions are not shown • 2 = equal and parallel dimensions are not shown, but both hook dimensions are shown • 3 = all dimensions are shown • 4 = hook dimensions are not shown • 5 = hook dimensions, or equal and parallel dimensions are not shown
PullOutShowUSHookDims	Defines if the US/NA style of dimension will be shown for hooks over 90 degrees. Options: • 0 = shows European hook dimension (= leg length, default) • 1 = shows US hook dimension (= straight length) for hooks >90 degrees See the image below for the difference between the US/NA (A) and European (B) hook dimension. 

Pour object and pour break properties in drawings

Use the options in **Pour Object Properties** and **Pour Break Properties** in general arrangement drawings to control the visibility of the pour objects and pour breaks in drawings.

NOTE Options and functionality related to pours are only available if pours are enabled in the model. Pours can be enabled in a model by setting the XS_ENABLE_POUR_MANAGEMENT (page 246) advanced option to

`TRUE`. In the default environment, pours are only enabled in the (Concrete) Contractor role.

Pour object properties

To open pour object properties in drawings:

- On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** and then click **Pour object...**
- In an open GA drawing, double-click the drawing background and click **Pour object...**
- In an open GA drawing, double-click the drawing view frame, and select **Pour object** from the options tree.
- In an open GA drawing, double-click a pour object.
- In an open GA drawing, go to **Quick Launch**, start typing `pour object properties`, and select **Drawing pour object properties** in the list.

The following table lists all pour object properties on all property levels (drawing, view, object).

Setting	Description
Content tab - **Hidden lines**	
Hidden lines on/off	Select the check box to display pour object hidden lines.
Own hidden lines on/off	Select the check box to display own hidden lines.
Content tab - **Additional marks**	
Edge chamfers on/off	Select **on** to display edge chamfers.
Fillet edges on/off	Select **on** to display fillet edges.
Appearance tab - **Visible lines**	
Color	Select the color of the visible pour object lines.
Type	Select the type of the visible pour object lines.
Appearance tab - **Hidden lines**	
Color	Select the color of the hidden pour object lines.
Type	Select the type of the hidden pour object lines.
Fill tab To add a fill for outer pour faces, use the **Pour faces** area, and to add a fill for cross sections in section views, use the **Sections** area.	

Setting	Description
Type	Defines the fill type. Clicking the button next to the list opens a preview of the hatch patterns. **Automatic** selects the fill type automatically from the hatch pattern schema files. **None** uses no fill.
Color	Defines the color for the fill. You can select a predefined color or use **Special** color that is not converted to black in printouts.
Background	Defines the background color for the fill. Background color selection is disabled for hardware hatches. Background color can be set for automatic hatches, but it has effect only if automatic hatch is not defined for the material in the hatch pattern schema file.
Scale	**Automatic** scales and rotates the fill automatically. **Custom** allows you to select scaling and rotation manually. **Scaling in direction x** and **Scaling in direction y** define the scales in x and y direction. **Keep ratio of x and y** retains the relative proportions in the hatch pattern. **Angle** rotates the fill. For example, **Angle** 0.0 is for horizontal and **Angle** 90.0 for vertical.

Pour break properties

To open **Pour Break Properties**:

- On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** and then click **Pour breaks...**
- In an open GA drawing, double-click the drawing background and click **Pour breaks...**

- In an open GA drawing, double-click the drawing view frame, and select **Pour break** from the options tree.
- In an open GA drawing, double-click a pour break.
- In an open GA drawing, go to **Quick Launch**, start typing `pour break properties`, and select **Drawing pour break properties** in the list.

Option	Description
Content tab	
Visibility	Defines if the pour breaks are visible (**Visible**) or not (**Not visible**). This setting is not available in object level properties.
Hidden lines	Select the check box to display pour break hidden lines.
Appearance tab - **Visible lines**	
Color	Select the color of the visible pour break lines.
Type	Select the type of the visible pour break lines.
Appearance tab - **Hidden lines**	
Color	Select the color of the hidden pour break lines.
Type	Select the type of the hidden pour break lines.

Placement properties for marks, dimensions, notes, texts and symbols in drawings

Use the settings in the **Placing** dialog boxes for annotation objects (dimensions, marks, notes, texts, dimensions, symbols) to control how the annotation objects are placed in a drawing.

You can set automatic placement properties for dimensions and marks before you create a drawing. In an open drawing, you can modify the placement properties of marks, notes, texts, symbols and dimensions.

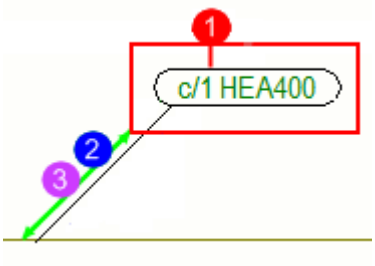
To open the **Placing** properties in an open drawing:

- In an open drawing, on the **Drawing** tab, click **Properties** and then click **Text**, **Note**, **Symbol**, **Dimension** or one of the mark types. After that, click the **Place...** button.
- In an open drawing, double-click a text, mark, symbol, note or dimension. After that, click the **Place...** button.

Option	Description
Search margin	Defines the empty margin that you want to have around the the annotation objects. Note that if you use a high **Search margin** value, the mark placement does not work properly.
Minimum distance	Defines the minimum distance of the mark, weld mark, dimension or another annotation object from the part. Note that if you use a high **Minimum distance** value, the mark placement does not work properly.
Maximum distance	Defines the maximum distance of the mark, dimension or another annotation object from the part.
Quarter	For marks and manually added annotation objects. Defines the areas Tekla Structures searches for a space to place the annotation object. Weld placement depends on the welding direction. Welds can only be placed in certain sectors, so the **Quarter** options are not available. However, this option is available for manually added welds in the final drawing.
Placing	**free** allows Tekla Structures to search for the first suitable location for the mark, dimension, weld or another annotation object. **fixed** allows you to place the mark, dimension, weld or another annotation object in any location. When you use the option **fixed**, the annotation object stays where it is even though you update the drawing, whereas with **free**, Tekla Structures tries to find the optimal place for the annotation object.

Option	Description
Direction	Available only for manual dimensions. Defines the side of the dimensioned object where Tekla Structures places dimensions. This setting affects the **free** setting.

The following image illustrates the search margin, minimum distance and maximum distance of a mark:



(1) Search margin

(2) Minimum distance

(3) Maximum distance

Model weld properties in drawings

You can select which model welds are visible in drawings and drawing views, and set the weld color and line type.

To go to weld properties:

- On the **Drawings & reports** tab, click **Drawing properties** , select the drawing type and navigate to weld properties.
- In an open drawing, double-click the drawing view frame, and click **Weld** in the options tree.
- In an open drawing, double-click the drawing background, and navigate to weld properties.
- Double-click a weld in an open drawing.
- In **Quick Launch**, start typing `weld properties`, and then select **Drawing weld properties** from the list.

The following table lists all weld settings on all property levels (drawing, view, object). All weld properties do not contain all of the settings listed below.

Setting	Description
Content tab - **Visibility**	

Setting	Description
Welds **Welds in sub-assemblies**	**Not visible** does not display any welds in the selected view/drawing. **Site weld visible** displays only site welds in the view/drawing. **Workshop weld visible** displays only workshop welds in the view/drawing **Both visible** displays both site welds and workshop welds in the view/drawing.
Weld size limit	Enter a weld size limit to filter welds of that size and smaller out of the drawing. This is useful when you only want to show non-typical welds in a drawing. To set whether the weld size is an exact or minimum value, use the advanced option XS_WELD_FILTER_TYPE (page 500). To filter out a standard weld type, use the advanced option XS_OMITTED_WELD_TYPE (page 335).
Content tab: **Representation**	
Representation	Select **Path** or **Outline**. You can also select whether to show **Hidden lines** or **Own hidden lines**. Weld solids are shown in drawings in the following cases: • Weld solids are shown in drawings for those weld types that have real solid support. Welds that have no real solid support are shown in the model with a hexagonal placeholder, and in drawings weld solids are not shown. • Welds that have custom cross-sections are also supported.
Appearance tab: **Visible lines**	
Color	Sets the color of the weld lines.
Type	Sets the type of the weld lines.
Appearance tab: **Hidden lines**	
Color	Sets the color of the hidden lines.
Type	Sets the type of the hidden lines.

Drawing sketch object properties

You can draw different kinds of sketch objects (graphical objects) in your drawings (lines, rectangles, polylines, polygons, arcs, circles, clouds) and use the sketch objects for highlighting particular spots of interest, for example. In sketch object properties you can check and change the appearance of the sketch objects.

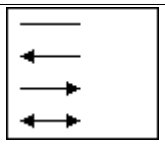
To open the properties of a sketch object, do one of the following:

- Go to the **Drawing** tab, hold down **Shift** and click any of the sketch object commands.

- When you have added a sketch object in a drawing, you can open its properties by double-clicking the object.
- You can also open sketch object properties through **Quick Launch**. For example, start typing `polygon properties`, and then select **Drawing polygon properties** in the list.

The available settings vary depending on the sketch object type. All settings are described below.

Setting	Description
Behind model objects	When set to **Yes**, places the graphical object behind model objects.
Line	
Type	Defines the line type of the object.
Color	Defines the color of the object lines.
Bulge or **Bulge for all lines**	Values 0 - 1.The bulge factor defines the curvature of the curved segments of objects using the following calculation: Arc height = Line length * Bulge factor Changing the bulge factor of a polyline or polygon modifies all the segments of that object.
Radius	Defines the radius of arcs and circles.
Arrow	
Position	Define the arrow position.

Setting	Description
	
Type	Define the arrow type for lines.
	Defines the arrow height.
	Defines the arrow length.
Fill: You can use fills (hatches) in rectangles, polygons, and circles.	
Type	Defines the fill (hatch) type to be used in the sketch object, for example, rectangle, polygon, or cloud. Clicking **Select...** shows the available hatch types. Note that there is no **Automatic** fill type available, like for part fills.
Color	Defines the color of the fill.
Background	Defines the background color of the fill. Background color selection is disabled for hardware hatches.
Scaling in direction x **Scaling in direction y**	Defines the fill scales in the x and y direction.
Keep ratio of x and y	Retains the relative proportions in the hatch pattern.
Angle	Rotates the fill. Angle 0.0 is for horizontal and 90.0 for vertical.
Offset	Moves the fill pattern inside the object in the x and y direction by the specified value.

Drawing grid and grid line properties

Use the grid and grid line properties to view and modify the grid settings in drawings.

- To set the automatic view-specific grid properties in single-part, assembly and cast unit drawings: On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type. Click **View creation** in the options

tree on the left, select the view and the properties that you want to change, and click **View properties**. Now click **Grid** in the options tree, and adjust the settings as required.

- To set the automatic drawing-specific grid properties in general arrangement drawing: On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type. Click **Grid...**, and adjust the settings as required.
- To modify the drawing-specific grid properties in a general arrangement drawing: Double-click the drawing background, click **Grid...**, and adjust the settings as required.
- To modify the view-specific grid properties: Double-click a drawing view frame and click **Grid** in the options tree, and adjust the settings as required.
- To modify the grid or grid line properties in an open drawing, double-click the grid or the grid line. Ensure that you have the correct selection switch active: To modify grids, use the selection switch , and to modify grid lines, use .
- To open the grid or grid line properties in an open drawing, go to **Quick Launch**, type `grid`, and select **Drawing grid properties** or **Drawing grid line properties** in the list.

Option	Description
Grid	**Visible** shows the grids. **Not visible** does not show the grids. This option is only available in drawing and view level properties. **Only grid labels visible** shows only the grid label and a short section of the grid line. The length of the displayed grid line depends on the value entered in the **Text placing** box. In the drawing level, this option is only available for general arrangement drawings. On the view and object level this option is available for all types of drawings.
Text placing	Select the sides to show the grid labels, and set the length of the grid line extension (the distance between the end of the grid line and the text).
Grid line: **Color** and **Type**	Define the grid line color and type.

Option	Description
Text: **Color**, **Height**, **Font** and **Frame**	Define the grid label text color, height, font and frame.

Other ways to adjust grids

Additionally, for example XS_DRAWING_GRID_LABEL_FRAME_FIXED_WIDTH (page 205), XS_DRAWING_GRID_LABEL_FRAME_LINE_WIDTH_FACTOR (page 206) and XS_GRID_TEXT_FONT (page 268) allow you to adjust the grid labels further. You can also customize the grid labels.

3.4 Report settings

Use the **Report** dialog box to check or change the report settings.

Option	Description
Report: Report templates	Lists all the available report templates.
Report:Titles in reports	Optional report titles. You can enter up to three report titles. All the titles are not used in every standard report. **Title1**, for example, is used to show phase information in the **Assembly_list** report.
Report: Browse	Use to change the folder where the report will be stored. By default, reports are stored in the current model folder.
Show	Shows the selected report.
Print	Prints the selected report.
Create from all	Creates a report from all objects in the model using the selected template.
Create from selected	Creates a report from the objects you have selected using the selected template.
Options: Show report	Defines how Tekla Structures displays reports. **On dialog** displays the report in a new window. **With associated viewer** displays the report in the associated program. For

Option	Description
	example, you can have Tekla Structures open all HTML reports in a web browser.
Options: Show created report	Defines whether or not the report is automatically shown on the screen after it has been created.

3.5 Analysis and design settings

This section provides information about the various analysis and design settings you can modify in Tekla Structures.

Click the links below to find out more:

Load group properties (page 792)

Load properties (page 793)

Load combination properties (page 800)

Analysis model properties (page 803)

Analysis part properties (page 809)

Analysis node properties (page 824)

Analysis rigid link properties (page 825)

Analysis bar position properties (page 827)

Analysis area position properties (page 827)

Analysis area edge properties (page 828)

Load group properties

Use the **Load Groups** dialog box to view, define, and modify the load group properties and to work with load groups.

Option	Description
Current	The **@** character identifies the current load group. When you create loads in the model, Tekla Structures adds them to the current load group. You can only define one load group as current. To change the current load group, select a load group and click **Set current**.

Option	Description
Name	Unique name of the load group. Use load group names to define the visibility and selectability of loads. For example, you can select, modify, or hide loads based on their load group.
Type	The type of a load group is the type of action that causes the loads. Actions causing loads are building code specific and depend on the load modeling code selected in **File menu** --> **Settings** --> **Options** --> **Load modeling** --> **Current code**. Most building codes use some or all of the following actions and load group types: • Permanent, dead, and/or prestressing loads • Live, imposed, traffic, and/or crane loads • Snow loads • Wind loads • Temperature loads • Accidental and/or earthquake loads • Imperfection loads
Direction	The direction of a load group is the global direction of the action that causes the loads. Individual loads in a load group retain their own magnitudes in the global or local x, y, and z directions. The load group direction affects which loads Tekla Structures combines in a load combination: • z direction groups are combined with both x and y direction groups. • x or y direction groups are **not** combined with each other.
Compatible	A number that identifies all the load groups that are compatible with each other.
Incompatible	A number that identifies all the load groups that are incompatible with each other.
Color	The color that Tekla Structures uses to show the loads in the group.

Load properties

This section provides information about the properties of specific loads.

Use the load properties dialog boxes to view, define, and modify the load properties. Each load type has its own properties dialog box.

Click the links below to find out more:

Point load properties (page 794)
Line load properties (page 794)
Area load properties (page 795)
Uniform load properties (page 796)
Temperature load properties (page 796)
Wind load properties (page 797)
Load panel settings (page 798)

Point load properties

Use the **Point Load Properties** dialog box to view and modify the properties of a point load or a bending moment. The file name extension of a point load properties file is `.lm1`.

Option	Description
Load group name	The load group to which the load belongs. To view load group properties or to create a new load group, click **Load groups**.
Magnitude tab	Load magnitudes in the x, y, and z directions of the work plane.
Load attachment	Indicates if the load is attached to a part.
Load-bearing parts	Parts to which the load is applied, or not applied, on the basis of part names or selection filters.
Bounding box of the load	Dimensions of the bounding box in the x, y, and z directions.
Load panel tab	See Load panel settings (page 798).

Line load properties

Use the **Line Load Properties** dialog box to view and modify the properties of a line load or a torsional moment. The file name extension of a line load properties file is `.lm2`.

Option	Description
Load group name	The load group to which the load belongs. To view load group properties or to create a new load group, click **Load groups**.
Magnitude tab	Load magnitudes in the x, y, and z directions of the work plane.
Load form	Defines how the load magnitude varies along the loaded length.
Load attachment	Indicates if the load is attached to a part.
Load-bearing parts	Parts to which the load is applied, or not applied, on the basis of part names or selection filters.
Bounding box of the load	Dimensions of the bounding box in the x, y, and z directions.
Distances	Offsets from the load end points, used to shorten or lengthen the loaded length. To shorten the loaded length, enter positive values for **a** and **b**. To lengthen the loaded length, enter negative values.
Load panel tab	See Load panel settings (page 798).

Area load properties

Use the **Area Load Properties** dialog box to view and modify the properties of an area load. The file name extension of an area load properties file is `.lm3`.

Option	Description
Load group name	The load group to which the load belongs. To view load group properties or to create a new load group, click **Load groups**.
Magnitude tab	Load magnitudes in the x, y, and z directions of the work plane.
Load form	Defines the shape of the loaded area.
Load attachment	Indicates if the load is attached to a part.
Load-bearing parts	Parts to which the load is applied, or not applied, on the basis of part names or selection filters.
Bounding box of the load	Dimensions of the bounding box in the x, y, and z directions.
Distances	Offset used to enlarge or reduce the loaded area. To enlarge the loaded area, enter a positive value for **a**. To reduce the loaded area, enter a negative value.

Option	Description
Load panel tab	See Load panel settings (page 798).

Uniform load properties

Use the **Uniform Load Properties** dialog box to view and modify the properties of a uniform load. The file name extension of a uniform load properties file is `.lm4`.

Option	Description
Load group name	The load group to which the load belongs. To view load group properties or to create a new load group, click **Load groups**.
Magnitude tab	Load magnitudes in the x, y, and z directions of the work plane.
Load attachment	Indicates if the load is attached to a part.
Load-bearing parts	Parts to which the load is applied, or not applied, on the basis of part names or selection filters.
Bounding box of the load	Dimensions of the bounding box in the x, y, and z directions.
Distances	Offset used to enlarge or reduce the loaded area.
Load panel tab	See Load panel settings (page 798).

Temperature load properties

Use the **Temperature Load Properties** dialog box to view and modify the properties of a temperature load or a strain. The file name extension of a temperature load properties file is `.lm6`.

Option	Description
Load group name	The load group to which the load belongs. To view load group properties or to create a new load group, click **Load groups**.
Temperature change for axial elongation	Temperature change in the part.
Temperature differential from side to side	Difference in temperature between the left side and the right side of a part.
Temperature differential from top to bottom	Difference in temperature between the top surface and the bottom surface of a part.

Option	Description
Initial axial elongation	Axial strain of a part. A positive value indicates elongation, a negative value indicates shrinkage.
Load attachment	Indicates if the load is attached to a part.
Load-bearing parts	Parts to which the load is applied, or not applied, on the basis of part names or selection filters.
Bounding box of the load	Dimensions of the bounding box in the x, y, and z directions.

Wind load properties

Use the **Wind Load Generator (28)** dialog box to view and modify the properties of wind loads.

To select or modify existing wind loads in the model as a group, use the **Select components** switch .

Option	Description
Wind load direction	The main direction of the wind. The options are: • **Global X** • **Global -X** • **Global Y** • **Global -Y** • **Global X, -X, Y, -Y** (for all directions)
Nominal wind pressure	The nominal value of wind pressure.
Top level	The highest level of the wind loads.
Bottom level	The lowest level of the wind loads.
Ground level	The level of the ground around the building.
Part names	Parts to which the load is applied, or not applied. See also Define load-bearing parts by name.
Front	The external exposure factors for the windward, leeward, and side walls. A positive value indicates pressure, a negative value indicates suction.
Left side	
Back	
Right side	
Internal	The internal exposure factor.

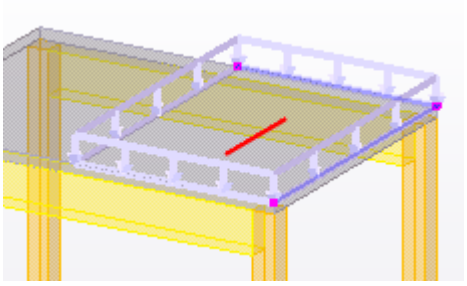
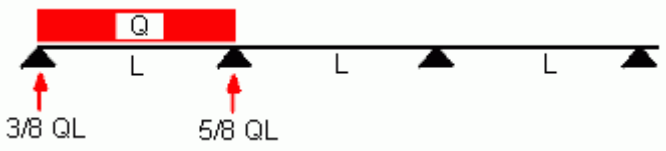
Option	Description
Z profile tab	The distribution of wind load along the height of the building, in terms of pressure factors. Starts from the ground level.
Global X, **Global Y**, **Global -X**, **Global -Y** tabs	A tab for each wind direction, where you can define zones for concentrated corner loads on each wall. Each zone is the height of the wall. Define the width of the zone using either dimensions or proportions. You can define up to five zones for each wall. Walls are numbered according to the order you pick points to indicate the shape of the building on the bottom level.

To select or modify individual existing wind loads in the model as separate area loads, use the **Select objects in components** switch and the **Area Load Properties** dialog box (page 795).

Load panel settings

Use the options on the **Load panel** tab in a load properties dialog box to modify the way Tekla Structures distributes the load.

Option	Description
Spanning	Defines the directions in which Tekla Structures distributes the load. • **Single** distributes the load only in the direction of the primary axis. • **Double** distributes the load along the primary and secondary axes.
Primary axis direction	Defines the direction of the primary axis using one of the following methods: • A value (1) in the **x**, **y**, or **z** box distributes the load in the corresponding global direction. • Values in multiple boxes distribute the load between the corresponding global directions. The values are the components of the direction vector. • Clicking **Parallel to part**, or **Perpendicular to part**, and then selecting a part in the model aligns the primary axis direction with the part.

Option	Description
	If **Spanning** is **Double**, you need to define the primary axis direction to be able to manually define the primary axis weight. To check the primary axis direction of a selected load in a model view, click **Show direction on selected loads**. Tekla Structures indicates the primary direction using a red line. 
Automatic primary axis weight	Defines whether Tekla Structures automatically weights the directions in load distribution. The options are: • **Yes**: Tekla Structures automatically calculates the load portions for the primary and secondary directions in proportion to the third power of the span lengths in these two directions. This means that the shorter the span, the bigger the proportion of the load. • **No**: You can enter the weight for the primary direction in the **Weight** box. Tekla Structures calculates the weight for the secondary direction by subtracting this value from 1.
Load dispersion angle	The angle by which the load is projected onto the surrounding parts.
Use continuous structure load distribution	Use for uniform loads on continuous slabs. Defines the distribution of support reactions in the first and last spans. The options are: • **Yes**: The distribution of support reactions is 3/8 and 5/8. 

Option	Description
	• **No**: The distribution of support reactions is 1/2 and 1/2.

Load combination properties

This section provides information about the settings that control the load combination process.

Click the links below to find out more:

- Load modeling code options (page 800)
- Load combination factors (page 800)
- Load combination types (page 801)

Load modeling code options

These are the load modeling codes available in Tekla Structures in **File menu** --> **Settings** --> **Options** --> **Load modeling** --> **Current code**:

Option	Description
Eurocode	European code
British	British code
AISC (US)	American Institute of Steel Construction, US code
UBC (US)	Uniform building code, US code
CM66 (F)	French code for steel structures
BAEL91 (F)	French code for concrete structures
IBC (US)	International building code, US code
ACI	American Concrete Institute's publication 318

Each of the available codes has a separate tab in the **Options** dialog box. The **Options** dialog box lists the partial safety factors in limit states and other combination factors for the code, based on load group types. For the Eurocode, you can also set the reliability class factor and the formula to be used in load combination.

See also

Load combination factors (page 800)

Load combination factors

In the load combination process, Tekla Structures uses partial safety factors and, for example, reduction factors on load groups to create load combinations.

The *partial safety factors* needed in the limit state design are:

- Unfavorable partial safety factor in the ultimate limit state (γ_{sup})
- Favorable partial safety factor in the ultimate limit state (γ_{inf})
- Unfavorable partial safety factor in the serviceability limit state (γ_{sup})
- Favorable partial safety factor in the serviceability limit state (γ_{inf})

Depending on the codes you use, you may need to use other combination factors. For example, the Eurocode contains three *reduction factors* (ψ_0, ψ_1, ψ_2). Reduction factors exclude the impractical effects of simultaneous loads.

You can use values for load combination factors that are building code specific or user-defined.

See also

Load combination types

You can perform several load combination types which vary according to the building code you use.

Use the **Load Combination Generation** dialog box, or the **Load Combination** dialog box, to select the load combination types you want to create. The options are:

Combination type	Description	Applies to
Ultimate limit state (ULS)	Combines load groups that occur persistently and transiently. Uses the partial safety factors of the ultimate limit state when combining loads.	Eurocode, British, AISC (US)
Serviceability limit state (SLS)	Combines load groups that occur quasi-permanently. Uses the partial safety factors of the serviceability limit state when combining loads.	Eurocode, AISC (US)
Serviceability limit state – Rare (SLS RC)	Combines load groups that occur quasi-permanently and rarely. Uses the partial safety factors of the serviceability limit state when combining loads.	Eurocode
Serviceability limit state – Quasi-permanent (SLS QP)	Combines load groups that occur quasi-permanently. Uses the partial	Eurocode

Combination type	Description	Applies to
	safety factors of the serviceability limit state when combining loads.	
Normal loads	Combines load groups and uses factors according to the French codes CM66 or BAEL91.	CM66, BAEL91
Extreme loads		CM66
Displacement loads		CM66
Accidental loads		CM66, Eurocode
Ultimate loads		BAEL91
Ultimate accidental loads		BAEL91
Earthquake loads	Combines load groups and uses factors according to the Eurocode.	Eurocode
Loads for public structures	Combines load groups according to the US IBC code (International Building Code).	IBC (US)
Loads for public structures with drifted snow		IBC (US)
Loads for non public structures		IBC (US)
Loads for non public structures with drifted snow		IBC (US)
Loads for public non concrete and masonry structures	Combines load groups according to the US UBC code (Uniform Building Code).	UBC (US)
Loads for public non concrete and masonry structures with drifted snow		UBC (US)
Loads for non concrete and masonry structures		UBC (US)
Loads for non concrete and masonry structures with drifted snow		UBC (US)
Loads for public concrete and masonry structures		UBC (US)
Loads for public concrete and masonry structures with drifted snow		UBC (US)

Combination type	Description	Applies to
Loads for concrete and masonry structures		UBC (US)
Loads for concrete and masonry structures with drifted snow		UBC (US)
ACI Table 1 - ACI Table 8	Combines load groups according to the ACI code (American Concrete Institute's publication 318).	ACI

See also

Analysis model properties

Use the **Analysis Model Properties** dialog box to define, view, and modify the properties of an analysis model. These properties apply to all parts in an analysis model.

Analysis model tab

Option	Description
Analysis application	The analysis application or format used in the analysis of the analysis model. To use the same application or format by default for other new analysis models, select the **Set as the default** check box. See also .
Analysis model name	A unique name for the analysis model. User-definable. For example, you can use a name that describes the portion of the physical model that you want to analyze. To define the export folder for the analysis model, click **Browse for export folder**.
Analysis model filter	Defines which objects to include in the analysis model, based on the list of available selection filters. See also .
Bracing member filter	Defines which of the included objects are considered to be braces. The analysis nodes of braces can move more freely than the ones of primary analysis parts when the analysis model is created.
Secondary member filter	Defines which of the included objects are considered to be secondary analysis parts. The nodes of

Option	Description
	secondary analysis parts can move more freely than the ones of primary analysis parts when the analysis model is created.
Analysis model content	Defines which objects are included in the analysis model. The options are: • **Selected parts and loads** Only includes selected parts and loads, and parts created by components, when they match the analysis model filter. To later add or remove parts and loads, use the **Add selected objects** or **Remove selected objects** button in the **Analysis & Design Models** dialog box. • **Full model** Includes all main parts and loads, except for parts whose analysis class (page 819) is **Ignore**. Tekla Structures automatically adds physical objects to the analysis model when they are created and when they match the analysis model filter. • **Floor model by selected parts and loads** Only includes selected columns, slabs, floor beams, and loads when they match the analysis model filter. Tekla Structures replaces columns in the physical model with supports. See also Analysis model content.
Use rigid links	Use to allow or prevent rigid links in the analysis model. The options are: • **Enabled** Rigid links are created if they are needed to connect analysis parts. • **Disabled, with keep axis: Default** No rigid links are created. The **Keep axis position** settings of the analysis parts are not changed.

Option	Description
	• **Disabled, with keep axis: No** No rigid links are created. The **Keep axis position** settings of the connected analysis parts are changed to **No**. If you use Tekla Structural Designer as the analysis application, you can use the **Enabled** option for concrete parts. The **Disabled, with keep axis: Default** option is automatically used for steel parts.
Analysis model rules	Click to create rules to define how Tekla Structures handles individual parts in the analysis model, and how parts are connected with each other in the analysis.
Curved beams	Defines whether beams are analyzed as curved beam or as straight segments. Select either: • **Split into straight segments** • **Use curved member** Use the advanced option XS_AD_CURVED_BEAM_SPLIT_ACCURACY_MM (page 60) in **File menu** --> **Settings** --> **Advanced Options** --> **Analysis & Design** to define how closely straight segments follow the curved beam.
Consider twin profiles	Defines whether twin profiles are considered as one part (**Enabled**) or as two parts (**Disabled**) in the analysis.
Member axis location	Defines the location of each analysis part in relation to the corresponding physical part. The options are: • **Neutral axis** The neutral axis is the analysis axis for all parts. The location of the analysis axis changes if the profile of the part changes. • **Reference axis (eccentricity by neutral axis)** The part reference line is the analysis axis for all parts. The location of the neutral axis defines axis eccentricity. • **Reference axis** The part reference line is the analysis axis for all parts.

Option	Description
	• **Model default** The analysis axis of each part is defined individually according to the analysis part properties. To define the axis location of specific parts, use the **Position** tab in the appropriate analysis part properties dialog box. If you select **Neutral axis**, Tekla Structures takes the part location and end offsets into account when it creates nodes. If you select either of the **Reference axis** options, Tekla Structures creates nodes at part reference points.
Member end release method by connection	Defines whether the support conditions of parts (**No**) or connections (**Yes**) are used.
Automatic update	Defines if the analysis model is updated according to the changes in the physical model. The options are: • **Yes - Physical model changes are considered** • **No - Physical model changes are ignored**
Model merging with analysis application	Only use with SAP2000 when changes occur in the Tekla Structures physical or analysis model that has already been exported to the analysis application. Defines whether the changed analysis model is merged with the previously exported model in the analysis application. The options are: • **Disabled** Models are not merged. Additions made in the analysis application to the previously exported model are lost. A new model is created every time you export the analysis model to the analysis application. • **Enabled** Models are merged. Additions made in the analysis application to the previously exported model are retained when you re-export the analysis model to the analysis application. The model in the analysis application is updated with the changes from Tekla Structures.

Analysis tab

Option	Description
Analysis method	Defines whether second order stresses are taken into consideration. The options are: • **1st order** Linear analysis method. • **P-Delta** A simplified second order analysis method. This method gives accurate results when deflections are small. • **Non-linear** Non-linear analysis method.
Maximum number of iterations	Tekla Structures repeats second order iteration until it reaches one of these values.
Accuracy of the iteration	
Modal analysis model	Select **Yes** to create a modal analysis model and to use modal analysis properties instead of static load combinations.

Job tab

Defines the job information in STAAD.Pro reports.

Output tab

Defines the contents of the STAAD.Pro analysis results file.

Seismic tab

Use the **Seismic** tab to define which building code to follow in the seismic analysis and the properties required by the seismic analysis. These properties vary depending on the code you select.

Option	Description
Type	The building code to use to generate seismic loads. The options are: • **None**: Seismic analysis not run. • **UBC 1997**: Uniform Building Code 1997 • **UBC 1994**: Uniform Building Code 1994 • **IBC 2000**: International Building Code 2000

Option	Description
	• **IS 1893-2002**: Indian Standard. Criteria for Earthquake Resistant Design of Structures • **IBC 2003**: International Building Code 2003 • **IBC 2006**: International Building Code 2006 • **IBC 2006 (ZIP)**: International Building Code 2006, with an option to add a ZIP code in the properties • **IBC 2006 (Longitude/Latitude)**: International Building Code 2006, with an option to add longitude and latitude information in the properties • **AIJ**: Japanese code • **Response spectrum**: Response spectrum specification
Seismic properties	Depending on the code you select, you can define various seismic properties.

Seismic masses tab

The loads and load groups included in the seismic analysis.

Modal analysis tab

Use the **Modal analysis** tab to define the properties required by the modal analysis.

Option	Description
Count of modes	The number of natural mode shapes in the structure.
Max frequency	The maximum natural resonant frequency of the structure.
Modal analysis masses	The loads and load groups included in the modal analysis.

Design tabs

Use the **Design** tabs for steel, concrete, and timber to define the codes and methods to use in structural design. The design options available vary depending on the material.

Option	Description
Design code	Design codes for different materials. The design code options available vary depending on the analysis application you use.

Option	Description
Design method	The material-specific principle used to compare stresses and material capacities. The options are: • **None** Tekla Structures only runs a structural analysis and creates data on stresses, forces, and displacements. Available for steel, concrete, and timber. • **Check design** Tekla Structures checks whether the structures fulfill the criteria in the design code (whether cross sections are adequate). Available for steel and timber. • **Calculate required area** Tekla Structures defines the required area of reinforcement. Available for concrete.
Design properties	The design code and method specific design properties of the analysis model that apply to all parts in the analysis model. When you select a design code and method for a material, Tekla Structures lists the design properties in the lower part of the **Design** tab. To change the value of a particular property, click on an entry in the **Value** column. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** . To change the design properties of a specific part, use the **Design** tab in the appropriate analysis part properties dialog box.

Analysis part properties

Use the options in a part's analysis properties dialog box (for example, **Beam Analysis Properties**) to define how Tekla Structures handles the part in the analysis. The settings you have available in the dialog box vary depending on the part type and analysis class. The table below lists all settings regardless of the part type and the analysis class.

Analysis tab

Use the **Analysis** tab to define the analysis properties of a part.

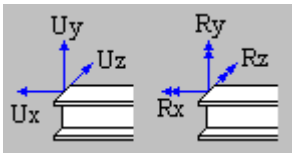
Option	Description
Class	Defines how the part is handled in the analysis. The selected **Class** defines which analysis properties are available. For example, plates have different properties from columns.
Filter (**Rigid diaphragm properties**)	Only available when the **Class** is **Contour plate - Rigid diaphragm** or **Slab - Rigid diaphragm**. Defines the filter used when filtering objects for a rigid diaphragm. Nodes that belong to a part matching the filter will be connected to the rigid diaphragm. For example, you can use a column filter to connect only column nodes to rigid diaphragms.
Built-up section mode	Indicates the role of the part in a built-up section that consists of a main part and one or more sub-parts. In the analysis, sub-parts are merged to the main part. The options are: • **Automatic** • **Not part of built-up section** Disconnects the part from a built-up section. • **Main part of built-up section** Always use to define the main part of a built-up section. • **Sub-part of built-up section** • **Beam sub-part of built-up section** Defines that the part is a part of the built-up section when the main part of the built-up section is a beam. • **Column sub-part of built-up section** Defines that the part is a part of the built-up section when the main part of the built-up section is a column.
Design group	Defines to which design group the part belongs. Used in optimization.
Automatic update	Defines if the analysis part is updated according to the changes in the physical model. The options are:

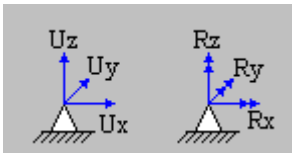
Option	Description
	• **Yes - Physical model changes are considered** • **No - Physical model changes are ignored**

Start releases tab, End releases tab

Use the **Start releases** and **End releases** tabs to define the support conditions and the degrees of freedom for the part ends.

The **Start releases** tab relates to the first part end (yellow handle), the **End releases** tab to the second part end (magenta handle).

Option	Description
Start or **End**	Defines which of the predefined or user-defined combinations for end conditions is used for part start or end. These are the predefined options: (Not available with Tekla Structural Designer) (Not available with Tekla Structural Designer) They automatically set the support condition and degrees of freedom. You can modify a predefined combination to suit your needs. If you do that, Tekla Structures indicates it with this option:
Support condition	Not available with Tekla Structural Designer. Defines the support condition. The options are: • **Connected** 

Option	Description
	Part end is connected to an intermediate analysis node (another part). Indicate degrees of freedom for the node. • **Supported**  Part end is the ultimate support for a superstructure (for example, the foot of a column in a frame). Indicate degrees of freedom for the support.
Rotation	Only available if **Support condition** is **Supported**. Defines whether the support is rotated. The options are: • **Not rotated** • **Rotated** If you select **Rotated**, you can define the rotation around the local x or y axis, or you can set the rotation by the current work plane by clicking **Set rotation by current work plane**.
Ux **Uy** **Uz**	Define the translational degrees of freedom (displacements) in the global x, y, and z directions. The options are: • **Free** • **Fixed** • **Spring** If you select **Spring**, enter the translational spring constant. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** .
Rx **Ry** **Rz**	Define the rotational degrees of freedom (rotations) in the global x, y, and z directions. The options are: • **Pinned** • **Fixed** • **Spring**

Option	Description
	• **Partial release** If you select **Spring**, enter the rotational spring constant. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** . Use **Partial release** to specify if the degree of connectivity is between fixed and pinned. Enter a value between 0 (fixed) and 1 (pinned).

Composite tab

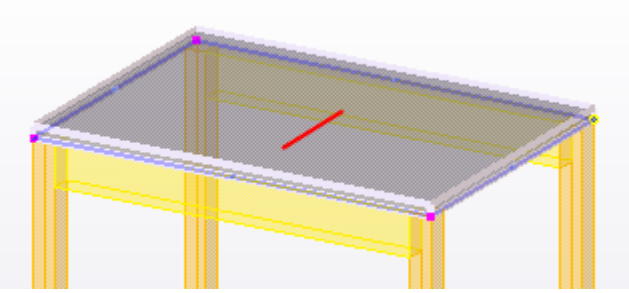
Use the **Composite** tab with STAAD.Pro to define the analysis properties of the slab in a composite beam.

Option	Description
Composite beam	Defines whether the composition is a: • **Non-composite beam** • **Composite beam** • **Automatic composite beam**
Material	Defines the material of the slab.
Thickness	Defines the thickness of the slab.
Effective slab width	Defines if the effective slab width is calculated automatically or based on the values you enter. You can define different values for the left and right side of the beam. Automatic values are calculated in relation to the span length.

Spanning tab

Use the **Spanning** tab to define the analysis and load distribution properties of a one-way or two-way slab system.

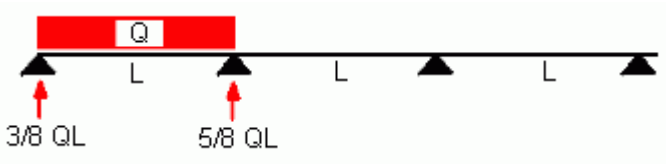
Option	Description
Spanning	Defines in which directions the part carries loads. The options are: • **Single** spanning plate carries loads in the direction of the primary axis. Beams or columns parallel to the spanning direction are not connected to the part, and will not carry loads from the part. • **Double** spanning part carries loads along the primary and secondary axes. Beams or columns in both directions will carry loads from the part.

Option	Description
Primary axis direction	Defines the direction of the primary axis in one of the following ways: • Enter 1 in the box (**x**, **y**, or **z**) which is parallel to the primary axis direction. • Enter values in multiple boxes to define the components of a direction vector. • Click **Parallel to part**, and then select a part in the model that is parallel to the direction. • Click **Perpendicular to part**, and then select a part in the model that is perpendicular to the direction. To check the primary spanning direction of a selected part in a model view, click **Show direction on selected members**. Tekla Structures indicates the primary direction using a red line. 

Loading tab

Use the **Loading** tab to include a part as loads in analysis models.

Option	Description
Generate self weight load	Analysis models include the part weight, for example a deck, as a load even if the part is not otherwise included in the analysis models. If the part is included in an analysis model, so is its self-weight. The option **No** works only with the analysis classes **Ignore** and **Rigid diaphragm**.
List boxes for additional loads	Enter slab live load or additional self-weight (screed, services) using three additional loads with a load group name and magnitude. The directions of these loads follow the direction of the load group to which they belong.
Part names	Use this filter to ensure that the area load from the slab is transferred to the correct parts, for example, beams supporting the slab. Typically, you would enter the beam name as the filter value.

Option	Description
Use continuous structure load distribution	Use to assign most of the load to the middle supports on continuous structures. 

Design tab

Use the **Design** tab in the analysis part properties dialog box to view and modify the design properties of an individual part in an analysis model. Design properties are properties which can vary, according to the design code and the material of the part (for example, design settings, factors, and limits).

Position tab

Use the **Position** tab to define the location and offsets of an analysis part.

Option	Description
Axis	Defines the location of the analysis part in relation to the corresponding physical part. The location of the analysis axis of a part defines where the part meets with other parts and where Tekla Structures creates nodes in analysis models. The options are: Neutral axis Reference axis (eccentricity by neutral axis) Reference axis Top left Top center Top right Middle left Middle center Middle right Bottom left Bottom center Bottom right Top plane Middle plane Bottom plane Left plane Right plane Middle plane (of left/right) If you select **Neutral axis**, Tekla Structures takes the part location and end offsets into account when it

Option	Description
	creates nodes. If you select either of the **Reference axis** options, Tekla Structures creates nodes at part reference points.
Keep axis position	Defines whether the axis position is kept or changed according to changes in the physical model. The options are: • **No** The axis is free to move when snapping end positions to nearby objects. Use this option for secondary members. • **Partial - keep in major direction** The axis is free to move partially, but the member is not moved in the major (stronger) direction of the part profile. • **Partial - keep in minor direction** The axis is free to move partially, but the member is not moved in the minor (weaker) direction of the part profile. • **Yes** The axis is not moved, but the end positions can move along the axis (thus extending or shortening the member). • **Yes - Keep end positions also** The axis and the end positions of the member are not changed.
Connectivity	Defines whether the member snaps or connects with rigid links to other members. The options are: • **Automatic** The member snaps or connects with rigid links to other members. • **Manual** The member does not snap or connect with rigid links to other members. Automatic connectivity to other members is created only if the member position matches the other member exactly.

Option	Description
Axis modifier X **Axis modifier Y** **Axis modifier Z**	Define whether the member location is bound to global coordinates, grid line, or neither. The options are: • **None** The member location is not bound. • **Fixed coordinate** The member location is bound to the coordinate you enter in the **X**, **Y**, or **Z** box. • **Nearest grid** The member is bound to the nearest grid line (the snap zone is 1000 mm).
Offset	Use to move the analysis part in the global x, y, and z directions.
Longitudinal offset mode	Defines whether the longitudinal end offsets **Dx** of the physical part are used from the physical part properties. The options are: • **Offsets are not considered** • **Only extensions are considered** • **Offsets are always considered**

Bar attributes tab

Use the **Bar attributes** tab in a frame object's (beam, column, or brace) analysis properties dialog box to define the properties of its analysis bars.

You can use the options on this tab when the analysis class of the analysis part is **Beam**, **Column**, or **Secondary**.

Option	Description
Start offset **End offset**	Calculate offsets to account for longitudinal eccentricity at the member end (resulting in a bending moment). These offsets have no effect on the topology on the analysis model. The offset value is only passed as a member attribute to the analysis.

Option	Description
Replacement profile name	Select a profile from the profile catalog. You can use different analysis profiles at the start and end of parts if the analysis application you use supports it. To use different profiles at part ends, enter two profiles separated by a pipe character, for example: `HEA120|HEA140` If the part is a built-up section in an analysis model, the name of the built-up section can be entered here. Any name can be entered, but if the name matches an existing catalog profile name, the physical properties of the section will be the same as the catalog profile properties.
Curved beam mode	Defines whether a beam is analyzed as a curved beam or as straight segments. The options are: • **Use model default** • **Use curved member** • **Split into straight segments** If you select **Use model default**, Tekla Structures uses the option selected from the **Curved beams** list in the **Analysis Model Properties** dialog box. Use the advanced option XS_AD_CURVED_BEAM_SPLIT_ACCURACY_MM (page 60) in **File menu** --> **Settings** --> **Advanced Options** --> **Analysis & Design** to define how closely straight segments follow the curved beam.
No. of split nodes	Use to create additional nodes or analyze a beam as straight segments, for example, a curved beam. Enter the number of nodes.
Split distances	To define additional nodes in the member, enter distances from the part starting point to the node. Enter distances, separated by spaces, for example: `1000 1500 3000`
Bar start number	Defines the start number for analysis bars.
Member start number	Defines the start number for analysis members.

Area attributes tab

Use the **Area attributes** tab in a plate's (contour plate, concrete slab, or concrete panel) analysis properties dialog box to define the properties of its analysis elements.

You can use the options on this tab when the analysis class of the analysis part is **Contour plate**, **Slab**, or **Wall**.

Option	Description
Element type	The shape of the elements.
Rotation of local XY	Defines the rotation of the local xy plane.
Element size	**x** and **y**: The approximate dimensions of the elements, in the local x and y direction of the plate. For triangular elements, the approximate dimensions of the bounding box around each element. **Holes**: The approximate size of the elements around openings.
Area start number	Defines the start number for the plate.
Simple area (ignore cuts etc)	Select **Yes** to create a simpler analysis model of the plate, where cuts and openings are not considered.
Smallest hole size to consider	Use to ignore small openings in the plate in the analysis. Enter the size of the bounding box around the opening.
Supported	Not available with Tekla Structural Designer. Use to define supports for a contour plate, concrete slab, or concrete panel. You can create supports for the bottom edge of a panel, for all edge nodes of a slab or plate, or for all nodes of a beam. For panels, the bottom edge can be inclined. The options are: • **No** No supports are created. • **Simply (translations)** Only translations are fixed. • **Fully** Both translations and rotations are fixed.

See also

Analysis class options and colors (page 819)

Analysis axis options (page 822)

Analysis class options and colors

Use the options in the **Class** list on the **Analysis** tab in a part's analysis properties dialog box to define how Tekla Structures handles the part in the analysis.

The option you select in the **Class** list determines which tabs are available in the analysis part properties (page 809) dialog box.

When the advanced option XS_AD_MEMBER_TYPE_VISUALIZATION (page 66) is set to `TRUE` (which is the default value), you can show the analysis class of parts using the following colors in the analysis model. You can also indicate the analysis classes using different colors in the physical model.

The analysis application you use may not support all of the following options. For example, the **Truss** options are not available with Tekla Structural Designer.

Option	Description	Color
Beam	Line object of two nodes. Part can take any load, including temperature.	Blue
Beam - Truss	Part can only take axial forces, not bending or torsion moments, or shear forces.	Bright green
Beam - Truss-Compression only	Part can only take compressive axial forces, not moments or shear forces. If this part goes into tension, it is ignored in the analysis.	Yellow
Beam - Truss-Tension only	Part can only take tensile axial forces, not moments or shear forces. If this part goes into compression, it is ignored in the analysis.	Pink
Beam - Ignore	Part is ignored in the analysis. Self-weight load is taken into account if you have set **Generate self weight load** to **Yes** on the **Loading** tab.	Part not shown in the model
Column	Vertical line object of two nodes. Modeled from bottom to top. Part can take any load, including temperature.	Blue
Column - Truss	Part can only take axial forces, not bending or torsion moments, or shear forces.	Bright green
Column - Truss-Compression only	Part can only take compressive axial forces, not moments or shear forces. If this part goes into tension, it is ignored in the analysis.	Yellow
Column - Truss-Tension only	Part can only take tensile axial forces, not moments or shear forces. If this part goes into compression, it is ignored in the analysis.	Pink

Option	Description	Color
Column - Ignore	Part is ignored in the analysis. Self-weight load is taken into account if you have set **Generate self weight load** to **Yes** on the **Loading** tab.	Part not shown in the model
Bracing	Line object of two nodes. Part can take any load, including temperature. For parts whose analysis class is **Bracing**, **Keep axis position** is off by default.	Green
Bracing - Truss	Part can only take axial forces, not bending or torsion moments, or shear forces.	Bright green
Bracing - Truss-Compression only	Part can only take compressive axial forces, not moments or shear forces. If this part goes into tension, it is ignored in the analysis.	Yellow
Bracing - Truss-Tension only	Part can only take tensile axial forces, not moments or shear forces. If this part goes into compression, it is ignored in the analysis.	Pink
Bracing - Ignore	Part is ignored in the analysis. Self-weight load is taken into account if you have set **Generate self weight load** to **Yes** on the **Loading** tab.	Part not shown in the model
Secondary	Line object of two nodes. Part can take any load, including temperature. For parts whose analysis class is **Secondary**, **Keep axis position** is off by default. Secondary parts snap to nearest nodes instead of part end nodes.	Orange
Secondary - Ignore	Part is ignored in the analysis. Self-weight load is taken into account if you have set **Generate self weight load** to **Yes** on the **Loading** tab.	Part not shown in the model
Wall - Shell	Part can take any load, except temperature.	Aqua
Wall - Plate	Same as **Wall - Shell** but plate elements are used in the analysis application.	Aqua
Wall - Shear wall	Part can take lateral forces and vertical forces.	Aqua
Wall - Ignore	Part is ignored in the analysis. Self-weight load is taken into account if you have set **Generate self weight load** to **Yes** on the **Loading** tab.	Aqua
Slab - Shell	Part can take any load, except temperature.	Aqua

Option	Description	Color
Slab - Plate	Same as **Slab - Shell** but plate, membrane, or mat foundation elements are used in the analysis application.	Aqua
Slab - Membrane		
Slab - Mat foundation		
Slab - Rigid diaphragm	Only applies to parts parallel to a global xy plane. **Filter**: Nodes that belong to a part matching the filter will be connected with rigid links which together affect displacement. For example, you can use a column filter to connect only column nodes to rigid diaphragms.	Lilac
Slab - Ignore	Part is ignored in the analysis. Self-weight load is taken into account if you have set **Generate self weight load** to **Yes** on the **Loading** tab.	Part not shown in the model
Contour plate - Shell	Part can take any load, except temperature.	Aqua
Contour plate - Plate	Same as **Contour plate - Shell** but plate or membrane elements are used in the analysis application.	Aqua
Contour plate - Membrane		Aqua
Contour plate - Rigid diaphragm	Only applies to parts parallel to a global xy plane. **Filter**: Nodes that belong to a part matching the filter will be connected with rigid links which together affect displacement. For example, you can use a column filter to connect only column nodes to rigid diaphragms.	Lilac
Contour plate - Ignore	Part is ignored in the analysis. Self-weight load is taken into account if you have set **Generate self weight load** to **Yes** on the **Loading** tab.	Part not shown in the model

Analysis axis options

Use the options in the **Axis** list on the **Position** tab in a part's analysis properties dialog box to define the location of the analysis part in relation to the physical part.

Option	Description	Use for
Neutral axis	The neutral axis is the analysis axis for this part. The location of the	

Option	Description	Use for
	analysis axis changes if the profile of the part changes.	
Reference axis (eccentricity by neutral axis)	The part reference line is the analysis axis for this part. The location of the neutral axis defines the axis eccentricity.	
Reference axis	The part reference line is the analysis axis for this part.	
Top left	The analysis axis is located in the top left corner of the part.	Beam objects (beams, columns, braces)
Top center	The analysis axis is located in the top center point of the part cross section.	Beam objects
Top right	The analysis axis is located in the top right corner of the part.	Beam objects
Middle left	The analysis axis is located in the middle of the left side of the part.	Beam objects
Middle center	The analysis axis is located in the center point of the part cross section.	Beam objects
Middle right	The analysis axis is located in the middle of the right side of the part.	Beam objects
Bottom left	The analysis axis is located in the bottom left corner of the part.	Beam objects
Bottom center	The analysis axis is located in the bottom center point of the part cross section.	Beam objects
Bottom right	The analysis axis is located in the bottom right corner of the part.	Beam objects
Top plane	The analysis axis is bound to the top plane.	Plate objects (plates, slabs, panels)
Middle plane	The analysis axis is bound to the middle plane.	Plate objects
Bottom plane	The analysis axis is bound to the bottom plane.	Plate objects
Left plane	The analysis axis is bound to the left plane.	Plate objects
Right plane	The analysis axis is bound to the right plane.	Plate objects
Middle plane (of left/ right)	The analysis axis is bound to the middle plane of left/right.	Plate objects

Tekla Structures uses the options above for each part when you select **Model default** from the **Member axis location** list in the **Analysis Model Properties** dialog box.

If you select **Neutral axis**, Tekla Structures takes the part location and end offsets into account when it creates nodes. If you select either of the **Reference axis** options, Tekla Structures creates nodes at part reference points.

TIP You can also use keyboard shortcuts to move the selected analysis part in relation to the physical part.

See also

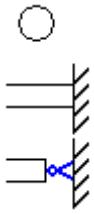
Analysis part properties (page 809)

Analysis model properties (page 803)

Analysis node properties

Use the **Analysis node properties** dialog box to view and modify the properties of a node in an analysis model.

To access the dialog box, double-click an analysis node.

Option	Description
Supports	Defines which support conditions are used for the node. The options are: • **Get supports from part(s)** The support conditions of a corresponding part end are used for the node. • **User-defined node supports** You can define the support conditions for the node. If you select **User-defined node supports**, you can select one of the following options: 

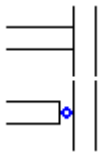
Option	Description
	These options automatically set the degrees of freedom for the node. You can modify a predefined combination to suit your needs. If you do that, Tekla Structures indicates it with this option:
Rotation	If you selected **User-defined node supports**, you can define the rotation of the node. The options are: • **Not rotated** • **Rotated** If you select **Rotated**, you can define the rotation, or you can set the rotation by the current work plane by clicking **Set rotation by current work plane**.
Ux **Uy** **Uz** **Rx** **Ry** **Rz**	Define the translational (U) and rotational (R) degrees of freedom (displacements and rotations) of the node in the global x, y, and z directions. The options are: • **Free** • **Fixed** • **Spring** If you select **Spring**, enter the spring constant. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** .

See also

Analysis rigid link properties

Use the **Analysis rigid link properties** dialog box to view and modify the end conditions of a rigid link.

To access the dialog box, double-click a rigid link.

Option	Description
Releases	Defines which releases are used for a rigid link start or end. The options are: • **Automatic releases (by rules)** • **User-defined releases**
Start or **End**	Defines which of the predefined or user-defined combinations for releases is used for a rigid link start or end. These are the predefined options:  These options automatically set the degrees of freedom. You can modify a predefined combination to suit your needs. If you do that, Tekla Structures indicates it with this option:
Ux **Uy** **Uz**	Define the translational degrees of freedom (displacements) in the global x, y, and z directions. The options are: • **Free** • **Fixed** • **Spring** If you select **Spring**, enter the translational spring constant. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** .

Option	Description
Rx **Ry** **Rz**	Define the rotational degrees of freedom (rotations) in the global x, y, and z directions. The options are: • **Pinned** • **Fixed** • **Spring** • **Partial release** If you select **Spring**, enter the rotational spring constant. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** . Use **Partial release** to specify if the degree of connectivity is between fixed and pinned. Enter a value between 0 (fixed) and 1 (pinned).
Local Y direction	Defines the local y direction of the rigid link. The options are the global x, y, and z directions. The local x direction is always the direction of the rigid link.

See also

Analysis bar position properties

Use the **Analysis Bar Position Properties** dialog box to view and modify the position of an analysis bar.

To access the dialog box, select an analysis bar, and then double-click a handle at an end of the analysis bar.

Option	Description
Offset mode	Defines whether the automatic (**Automatic offset**) or user-defined (**Manual offset**) offset values are used for the analysis bar end.
Offset	Defines the offset values in the global x, y, and z directions.

See also

Analysis area position properties

Use the **Analysis Area Position Properties** dialog box to view and modify the position of an analysis area.

To access the dialog box, select an analysis area, and then double-click a handle at an analysis area corner.

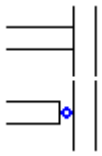
Option	Description
Offset mode	Defines whether the automatic (**Automatic offset**) or user-defined (**Manual offset**) offset values are used for the analysis bar end.
Offset	Defines the offset values in the global x, y, and z directions.

See also

Analysis area edge properties

Use the **Analysis Area Edge Properties** dialog box to view and modify the position and connectivity of an analysis area edge.

To access the dialog box, select an analysis area, and then double-click a handle at the mid-point of an analysis area edge.

Option	Description
Offset mode	Defines whether the automatic (**Automatic offset**) or user-defined (**Manual offset**) offset values are used for the analysis bar end.
Offset	Defines the offset values in the global x, y, and z directions.
Releases	Defines which of the predefined or user-defined combinations for releases is used for the analysis area edge. These are the predefined options:  These options automatically set the degrees of freedom. You can modify a predefined combination to suit your needs. If you do that, Tekla Structures indicates it with this option:

Option	Description
Ux **Uy** **Uz**	Define the translational degrees of freedom (displacements) in the global x, y, and z directions. The options are: • **Free** • **Fixed** • **Spring** If you select **Spring**, enter the translational spring constant. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** .
Rx **Ry** **Rz**	Define the rotational degrees of freedom (rotations) of a member end in the global x, y, and z directions. The options are: • **Pinned** • **Fixed** • **Spring** • **Partial release** If you select **Spring**, enter the rotational spring constant. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** . Use **Partial release** to specify if the degree of connectivity is between fixed and pinned. Enter a value between 0 (fixed) and 1 (pinned).

See also

4 Predefined parametric profiles available in Tekla Structures

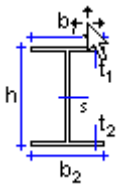
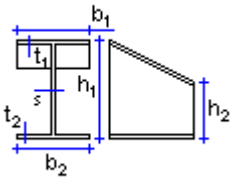
The below predefined parametric profiles are available in Tekla Structures.

The profiles are listed in the same order as they appear in the profile catalog in the default environment.

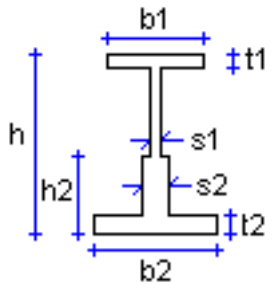
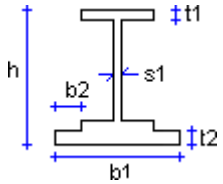
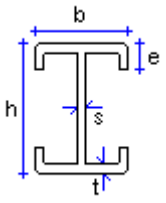
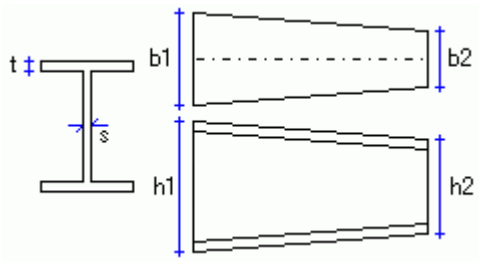
To change how the profiles are grouped in the profile catalog, modify the profile catalog rules.

You can download additional predefined profiles from Tekla Warehouse.

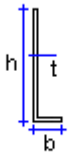
4.1 I profiles

	HIh-s-t*b (symmetric) HIh-s-t1*b1-t2*b2
	HIh1-h2-s-t*b HIh1-h2-s-t1*b1-t2*b2

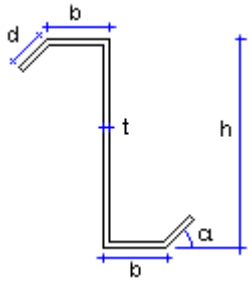
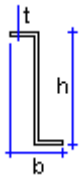
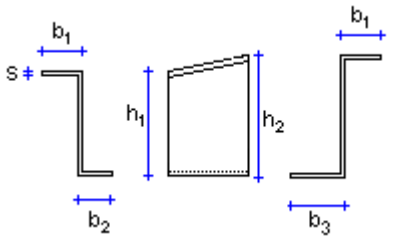
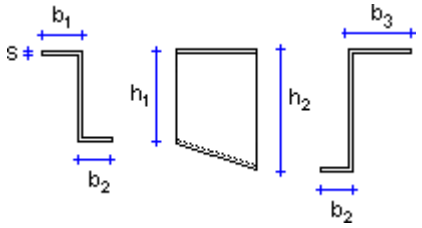
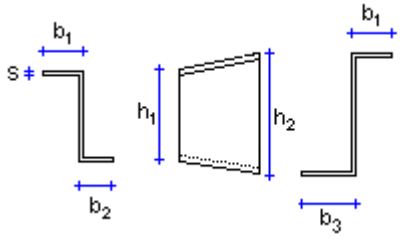
4.2 I beams (steel)

	I_BLT_Ah-b1-s1-t1*h2-b2-s2-t2
	I_BLT_B h*b1*t1*s-b2*t2
	I_HEMh*b*c*s*t
	I_VAR_Ah1-ht*b1-bt*s*t

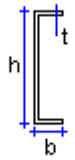
4.3 L profiles

	Lh*b*t

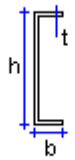
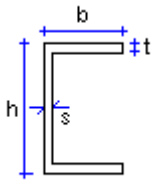
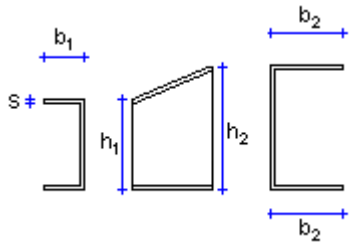
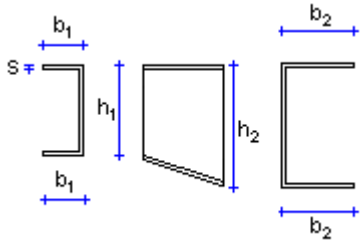
4.4 Z profiles

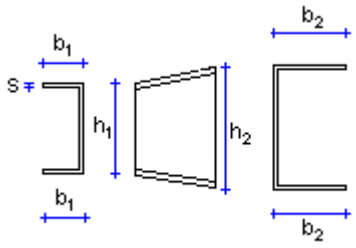
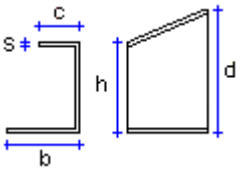
	BENTZ h*b*d*t[-a]
	Z h*b*t
	Z_VAR_A h1*b1*b2-s-h2*b3
	Z_VAR_B h1*b1*b2-s-h2*b3
	Z_VAR_C h1*b1*b2-s-h2*b3

4.5 U profiles

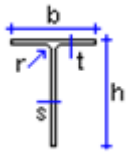
	Uh*b*t

4.6 C profiles

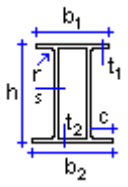
	Ch*b*t
	C_BUILTh*b*s*t
	C_VAR_Ah1*b1-s-h2*b2
	C_VAR_Bh1*b1-s-h2*b2

	C_VAR_Ch1*b1-s-h2*b2
	C_VAR_Dh-b-d-c-s

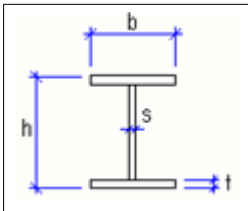
4.7 T profiles

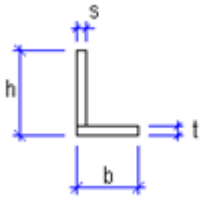
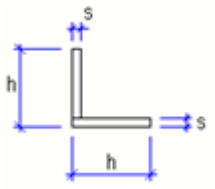
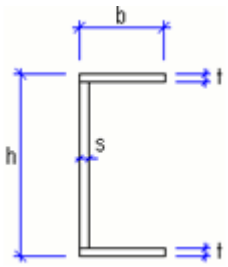
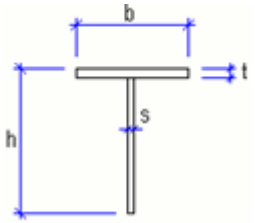
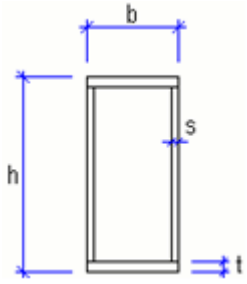
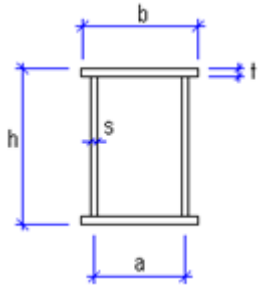
	Th-s-t-b

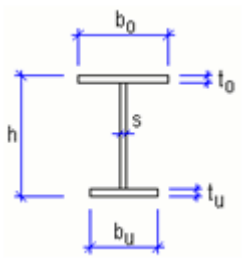
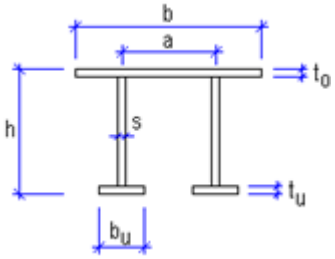
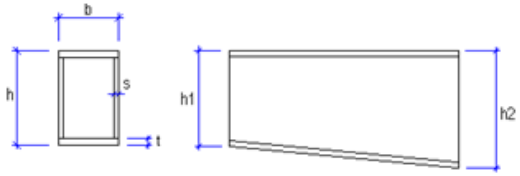
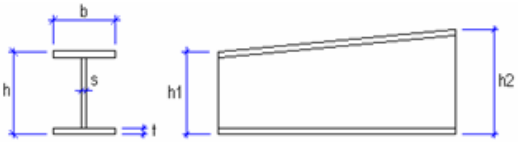
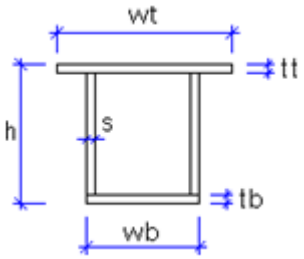
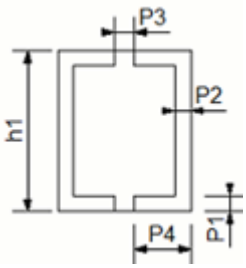
4.8 Welded box profiles

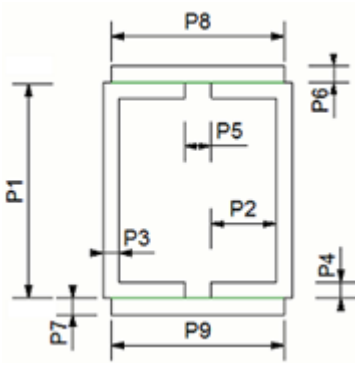
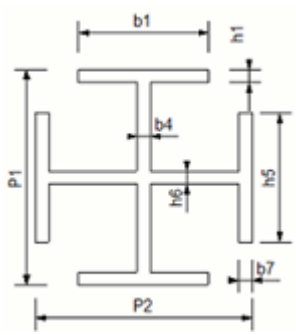
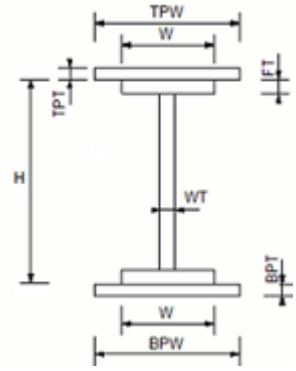
	HK h-s-t*b-c HKh-s-t1*b1-t2*b2-c

4.9 Welded beam profiles

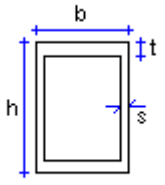
	B_WLD_A h*b*s*t

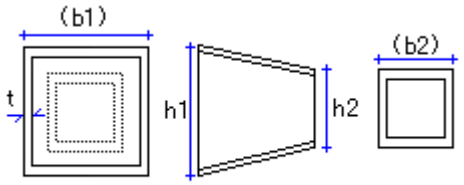
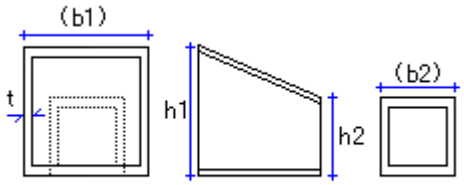
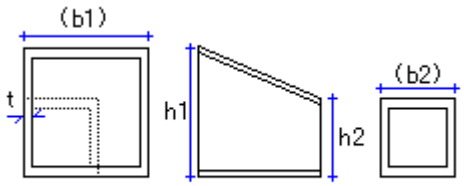
	B_WLD_B h*b*s*t
	B_WLD_C h*s
	B_WLD_D h*b*s*t
	B_WLD_E h*b*s*t
	B_WLD_F h*b*s*[t]
	B_WLD_G h*b*s*t*a

	B_WLD_H h*bo*bu*s*to*tu
	B_WLD_I h*bo*s*to*bu*tu*a
	B_WLD_J h1*h2*b*s*t
	B_WLD_K h1*h2*b*s*t
	B_WLD_L h*wt*wb*s*tt*tb
	B_WLD_M h1*p1*p2*p3*p4

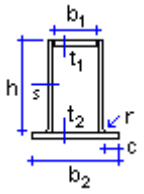
	B_WLD_N p1*p2*p3*p4*p5*p6*p7*p8*p9
	B_WLD_O b1*h1*b4*h5*b7*h6*P1*P2
	B_WLD_P W*H*FT*WT*TPT*TPW*BPT*BPW

4.10 Box profiles

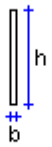
	B_BUILTh*b*s*t

	B_VAR_Ah1-h2*t
	B_VAR_Bh1-h2*t
	B_VAR_Ch1-h2*t

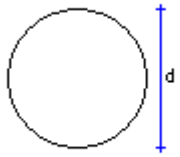
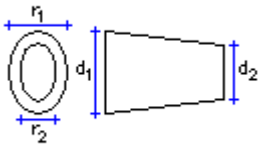
4.11 WQ profiles

	HQh-s-t1*t2*b2 HQh*s-t1*b1-t2*b2-c

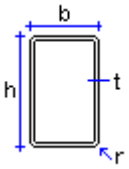
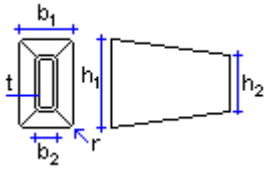
4.12 Rectangular sections

	PLh*b h=height b=thickness (smaller=b)

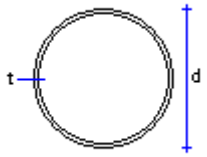
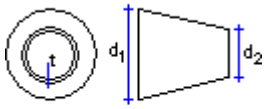
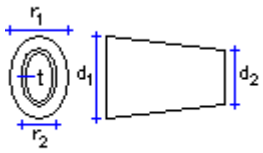
4.13 Circular sections

	Dd
	ELDd1*r1*d2*r2

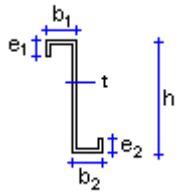
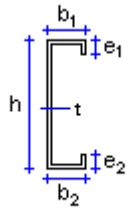
4.14 Rectangular hollow sections

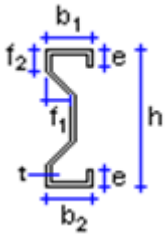
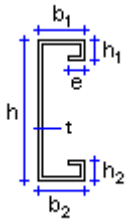
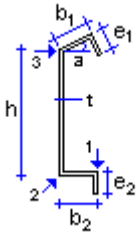
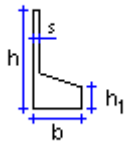
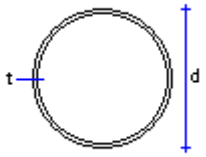
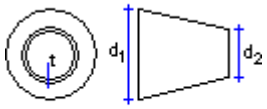
	Ph*t (symmetric) Ph*b*t
	Ph1*b1-h2*b2*t

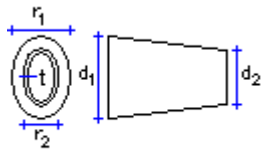
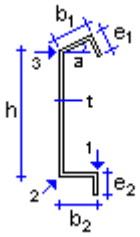
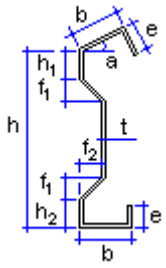
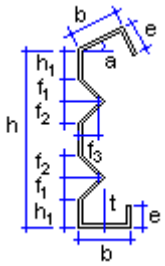
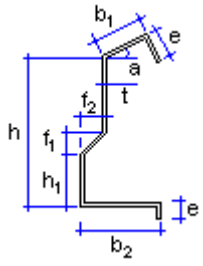
4.15 Circular hollow sections

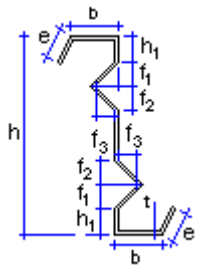
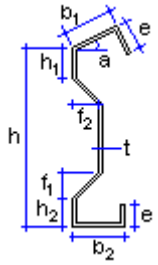
	PDd
	PDd1*d2*t
	EPDd1*r1*d2*r2*t

4.16 Cold rolled profiles

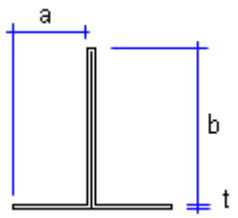
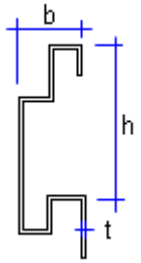
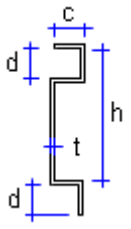
	ZZh-t-e-b (symmetric) ZZh-t-e1-b1-e2-b2
	CCh-t-e-b (symmetric) CCh-t-e1-b1-e2-b2

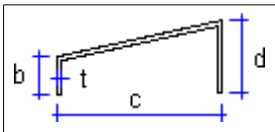
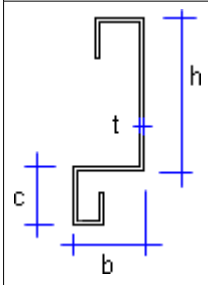
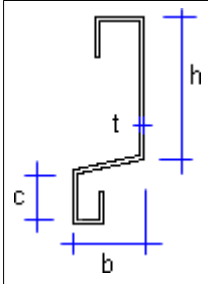
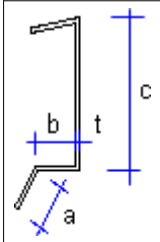
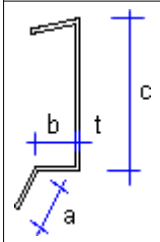
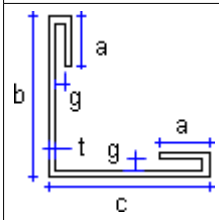
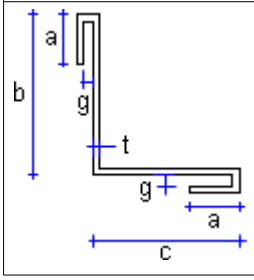
	CW h-t-e-b-f-h1 (symmetric) CW h-t-e1*b1-f1-f2-e2*b2
	CUh-t-h1-b-e (symmetric) CUh-t-h1-b1-h2-b2-e
	EBh-t-e-b-a EBh-t-e1-b1-e2-b2-a Reference points: 1=right 2=left 3=top
	BFh-s-b-h1
	SPDd*t
	SPDd2*d2*t

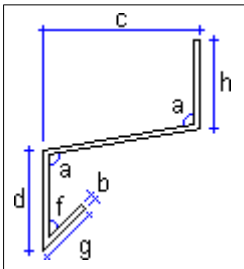
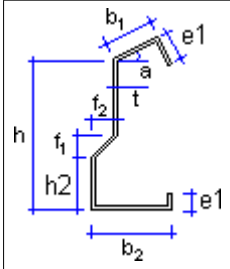
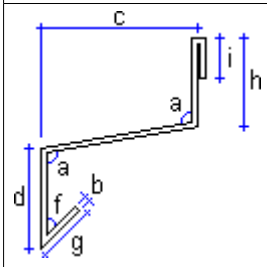
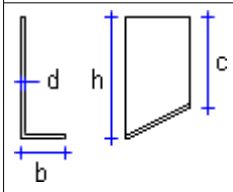
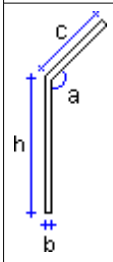
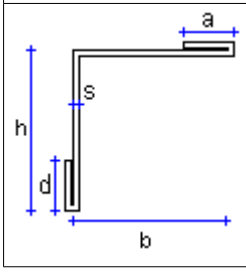
	ESPD d1-d2*t
	ECh-t-e-b-a ECh-t-e1-b1-e2-b2-a
	EDh-t-b-e-h1-h2-f1-f2-a
	EEh-t-e-b-f1-f3-h1-f2-a
	EFh-t-e-b1-b2-f1-f2/h1-a

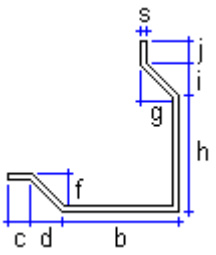
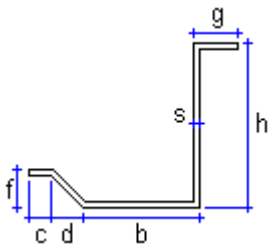
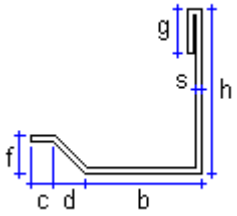
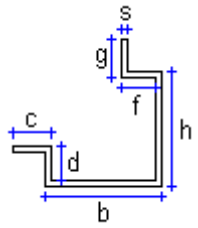
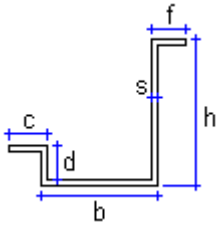
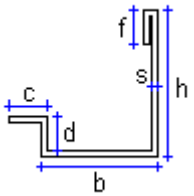
	EZh-t-e-b-f1-f3-h1-f2-a
	EWh-t-e-b1-b2-f1-f2-h2-h1-a

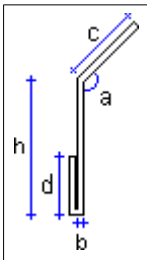
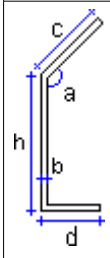
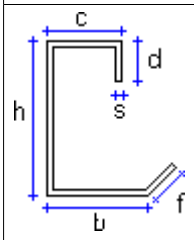
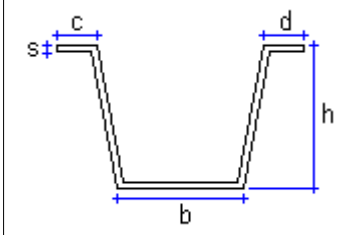
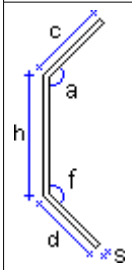
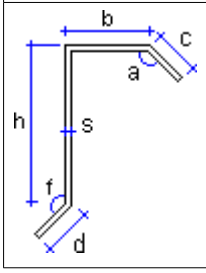
4.17 Folded plates

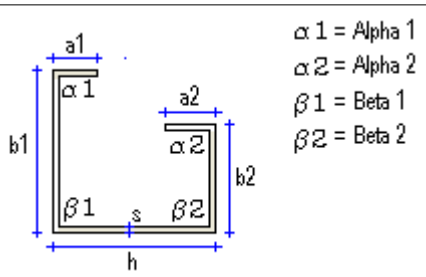
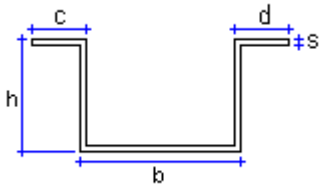
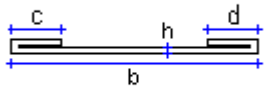
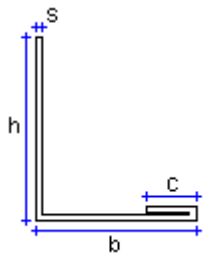
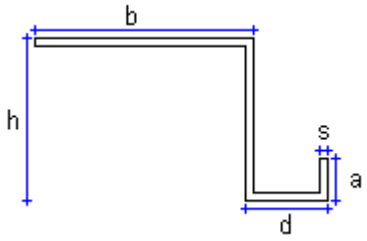
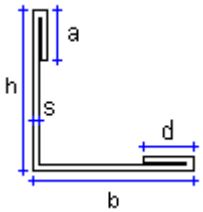
	FFLAa-b-t
	FPANBh-b-t FPANB_-b-t FPANBAh-b-t FPANBA_h-b-t
	FPANBBh-c-d-t

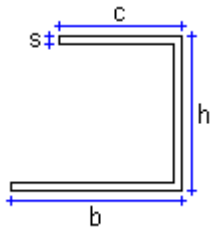
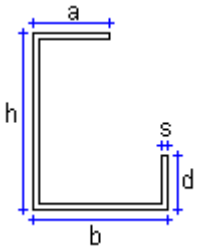
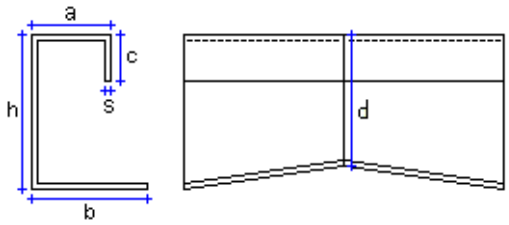
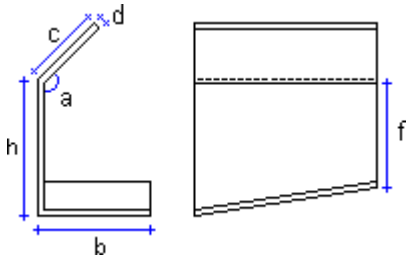
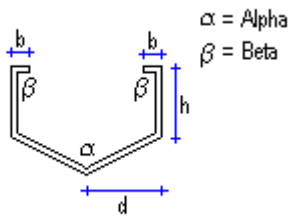
	FPANCVb-c-d-t
	FPANGh-b-c-t
	FPANGAh-b-c-t
	FPANJa-b-c-t
	FPANJa-b-c-t
	FPAN a-b-c-t-g
	FPANVVa-b-c-t-g

	FP_Ah-b-c-d-g
	FP_AAh*b2*t*a
	FP_Bh-b-c-d-g-i
	FP_BBh-b-d
	FP_Cb-h-c
	FP_CCh-b-a-d-s

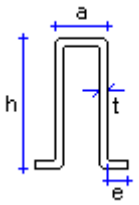
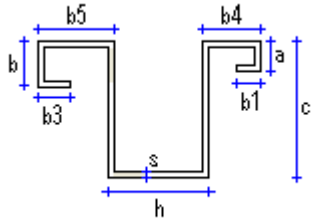
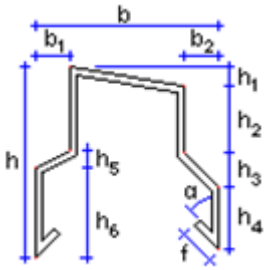
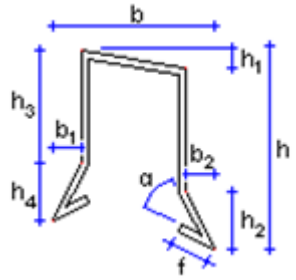
	FP_Db-h-c-d-f-g-i-j-s
	FP_Eb-h-c-d-f-g-s
	FP_Fb-h-c-d-f-g-s
	FP_Gb-h-c-d-f-g-s
	FP_Hb-h-c-d-f-s
	FP_Ib-h-c-d-f-s

	FP_Jb-h-c-d-a
	FP_Kb-h-c-d
	FP_Lb-h-c-d-f-s
	FP_Mb-h-c-d-s
	FP_Nb-h-c-d
	FP_Ob-h-c-d-s

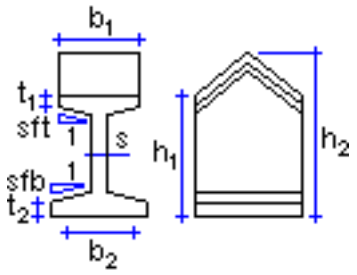
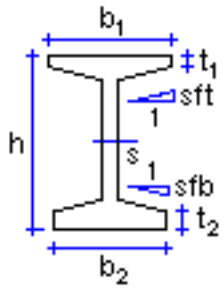
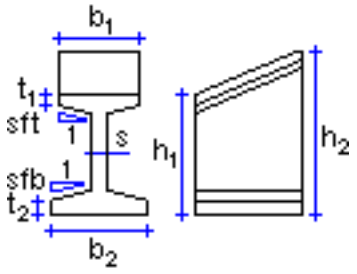
	FP_Pa1*a2*h-b1*b2-Alpha1-Alpha2-Beta1-Beta2-s
	FP_Qb-h-c-d-s
	FP_Rb-h-c-d
	FP_Sb-h-c-s
	FP_Tb-h-a-d-s
	FP_Ub-h-a-d-s

	FP_Vb-h-s-c
	FP_Wb-h-a-d-s
	FP_WWh-b-a-c-s
	FP_Yh-b-c-d
	FP_Zd-h-b-s-a-f

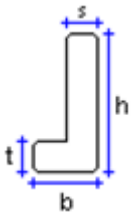
4.18 Hat profiles

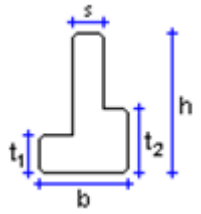
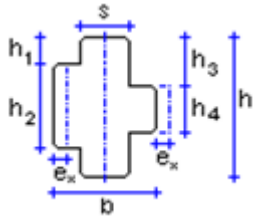
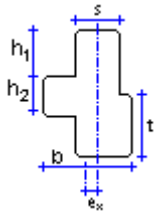
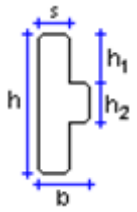
	HAT h*a*c*t
	HATCa-b-c-b1-h-b3-b4-b5-s
	HATAb1*h1*h2*h3*h4*h5*h6*b2*t*f*a*h*b
	HATBb*b1*b2*h*h1*h2*h3*h4*t*f*a

4.19 I beams (concrete)

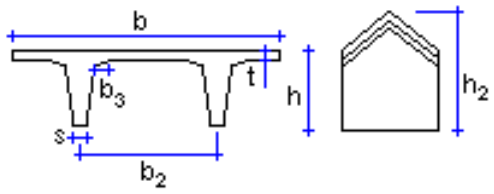
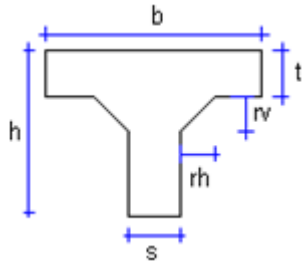
	HIIh1*b1*t1-h2-s-b2*t2[-sft[-sfb]]
	IIh*b1*t1-s-b2*t2[-sft[-sfb]]
	SIIh1*b1*t1-h2-s-b2*t2[-sft[-sfb]]

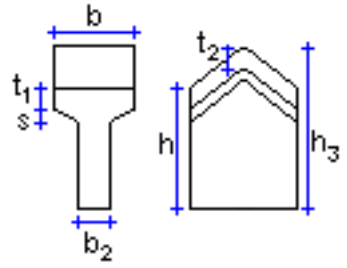
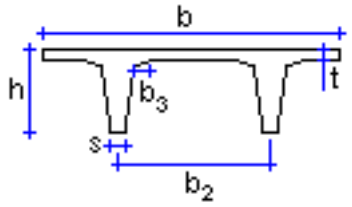
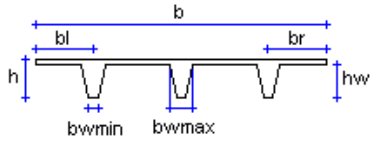
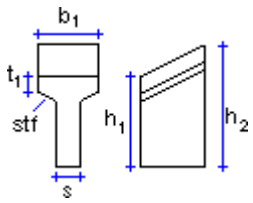
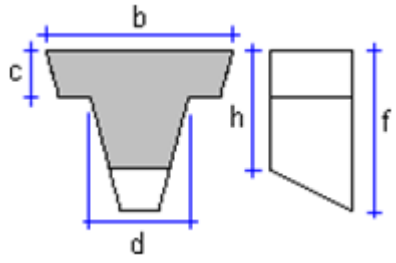
4.20 Ledger beams (concrete)

	RCLs*h-b*t

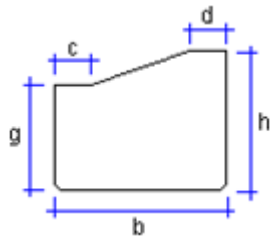
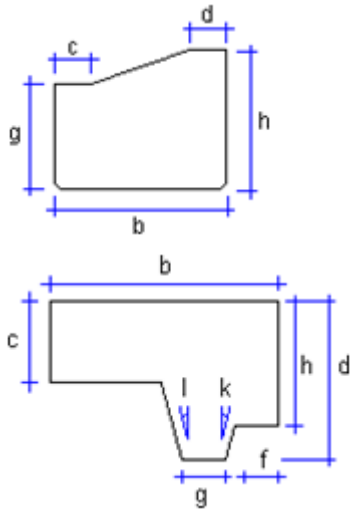
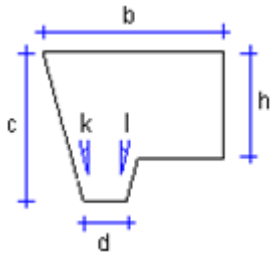
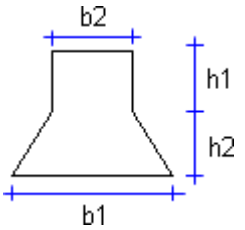
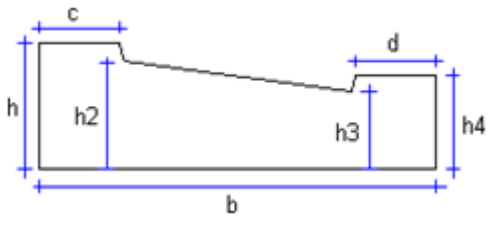
	RCDLs*h-b*t RCDLs*h-b*t1*t2
	RCDXs*h-b*h2*h1 RCDXs*h-b*h4*h3*h2*h1 RCDXs*h-b*h4*h3*h2*h1-ex
	RCXXs*h-b*t*h1-h2-ex
	RCXs*h-b*h2*h1

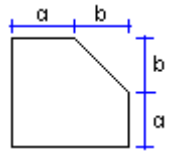
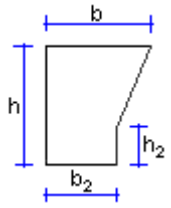
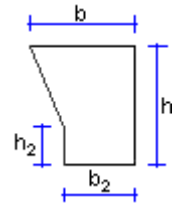
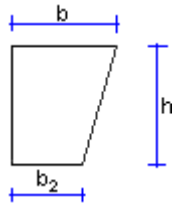
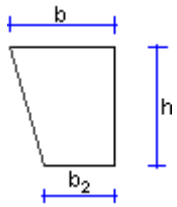
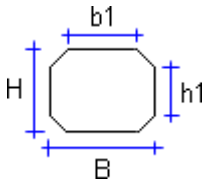
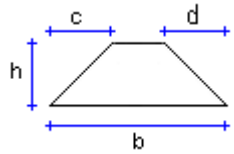
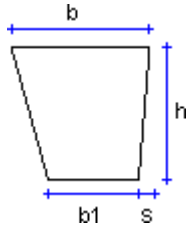
4.21 T profiles (concrete)

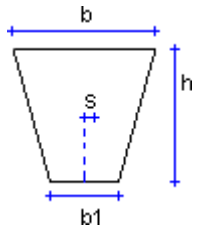
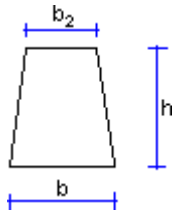
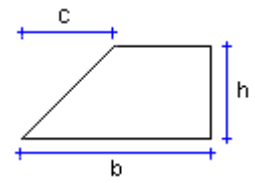
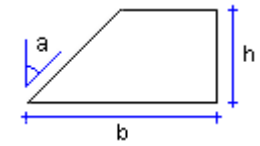
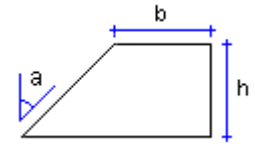
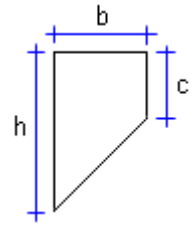
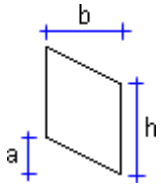
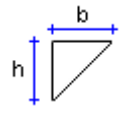
	HTTh*b-s-t-b2-h2
	TCh-b-t-s

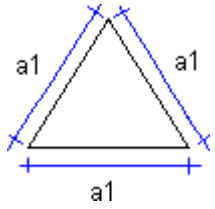
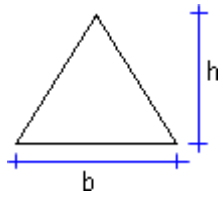
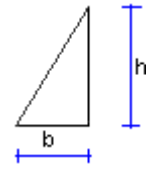
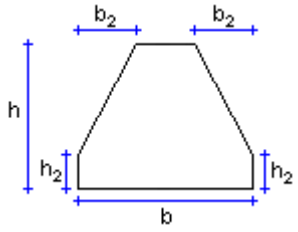
	TRIh*b-b2*t1-h3-t2
	TTh*b-s-t-b2
	TTTh*b-bl-br-hw-bwmin-bwmax
	T_VAR_Ah1*h2*s*b1*t1-sft
	T_VAR_Bh-b-c-d

4.22 Irregular beams (concrete)

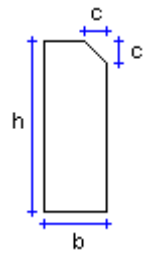
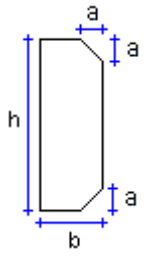
	IRR_Ab-h-g-c-d
	IRR_Bh-b-c-d-f-g
	IRR_Ch-b-c-d
	IRR_Db1*b2-h1*h2
	IRR_Eh-b-c-d-h2-h3-h4

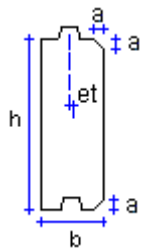
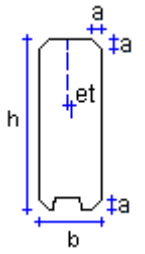
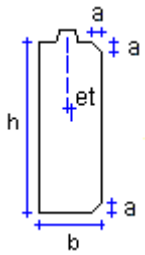
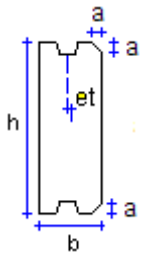
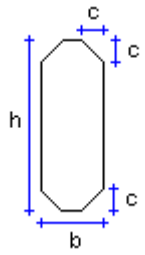
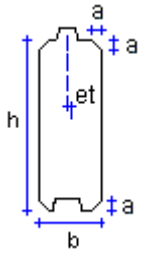
	IRR_Fa*b
	IRR_Gh*b*h2*b2
	IRR_Hh*b*h2*b2
	IRR_Ih*b*b2
	IRR_Jh*b*b2
	OCTB*b1-H*h1
	REC_Ah-b
	REC_Bh-b-b1

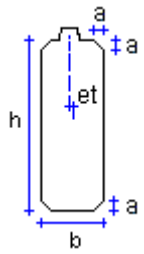
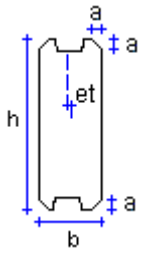
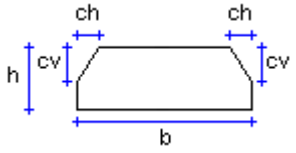
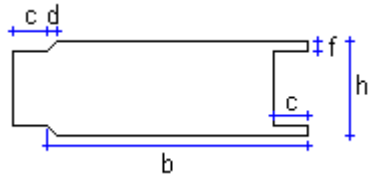
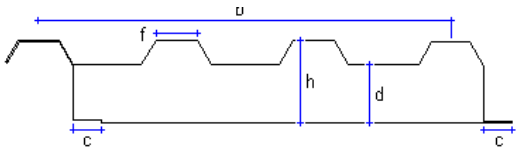
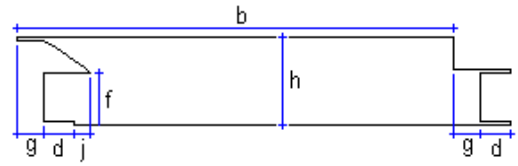
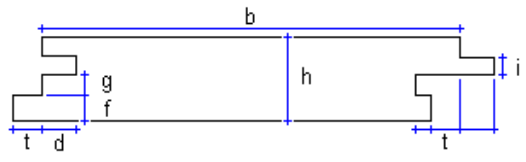
	REC_Ch-b-b1
	REC_Dh-b-b2
	REC_Eh-b
	REC_Fh-b
	REC_Gh-b
	REC_Hh-b
	REC_I a-b*h
	TRI_Ah-b

	TRI_Ba1
	TRI_Cb-h
	TRI_Dh*b
	TRI_Eb*h*h2*b2

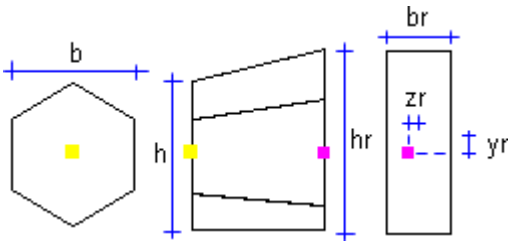
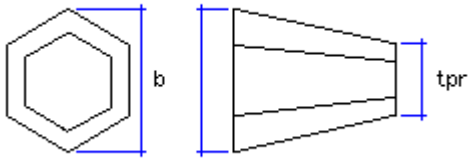
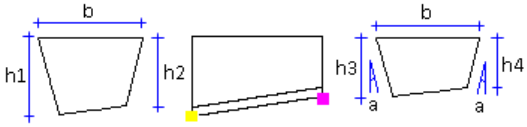
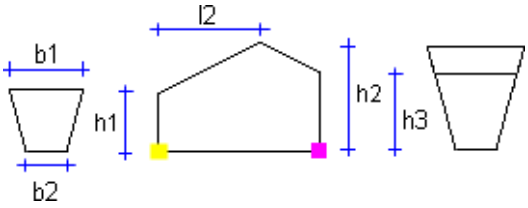
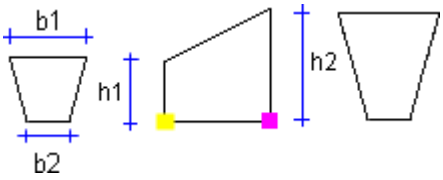
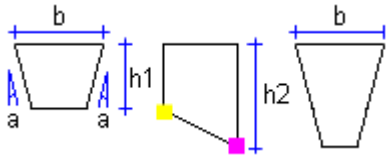
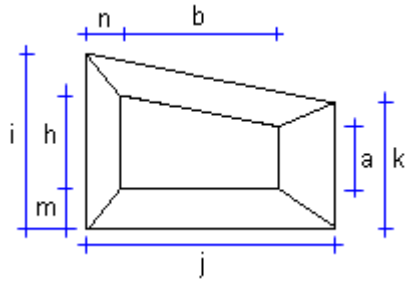
4.23 Panels

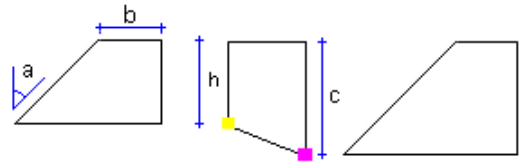
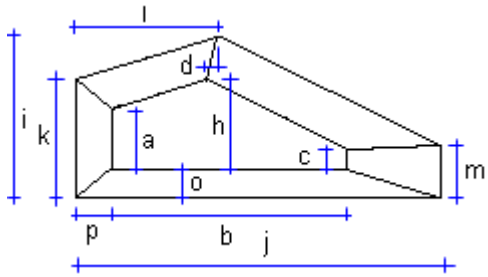
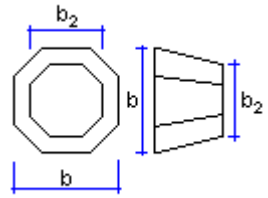
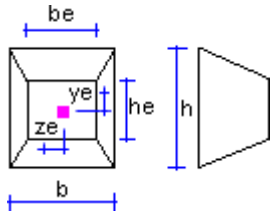
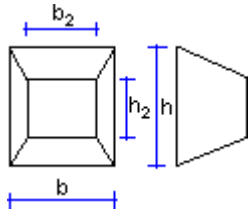
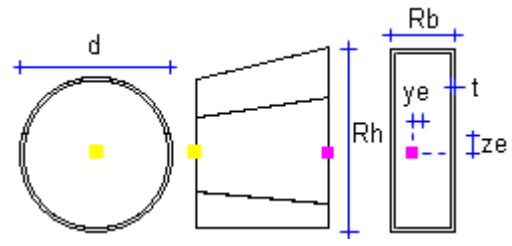
	PNL_Ah*b
	PNL_Bh*b

	PNL_Ch*b-a-ht*bt
	PNL_Dh*b-a-ht*bt
	PNL_Eh*b-a-ht*bt
	PNL_Fh*b-a-ht*bt
	PNL_Gh*b
	PNL_Hh*b-a-ht

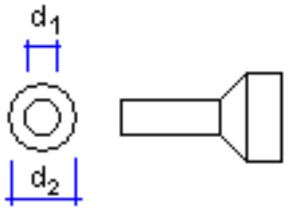
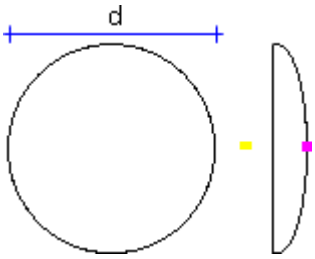
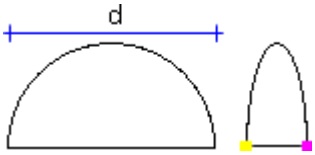
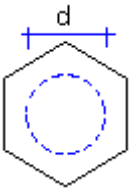
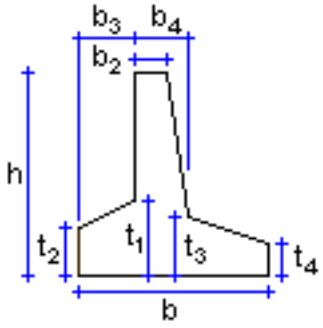
	PNL_Ih*b-a-ht*bt
	PNL_Jh*b-a-ht*bt
	PNL_Kh*b
	PNL_Lh-b-c-f
	PNL_Mh-b-c-f-d
	PNL_Nh-b-d-f-g-j
	PNL_Oh-b-d-f-g-i-t

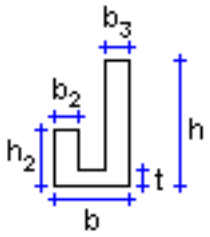
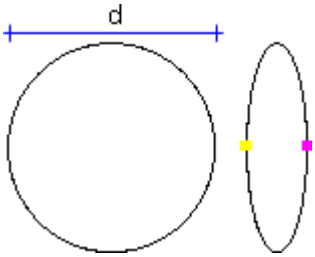
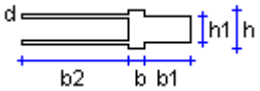
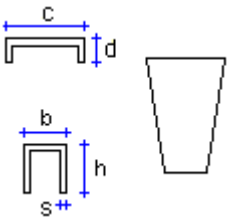
4.24 Variable cross sections

	HEXRECTh-b-br-hr
	HXGONb
	OBLINCLh1-h2-h3-h4-b
	OBLRIDh1*b1*b2-h2-h3-l2-l1
	OBLVAR_Ah1*b1*b2-h2
	OBLVAR_Bh1-h2-b
	OBLVAR_Ch-b-a-i-j-k-m-n

	OBLVAR_Dh-c-b
	OBLVAR_Eh-b-a-c-d-i-j-k-l-m-p-o
	OCTAGONb-b2
	PRMDASh*b-he*be PL_Vh*b-he*be
	PRMDh*b-h2*b2
	ROUNDRECTd-Rb*Rh-t*ye-ze

4.25 Others

	BLKSd1-d2
	CAPd
	HEMISPHERd
	NUT_Md
	RCRWh*b-b2*b3-b4-t1*t2-t3*t4

	SKh*b-h2-t-b2-b3
	SPHEREd
	STBb-h-h1-b1-b2-d
	STEPh-b*h1-b1-s

5 Steel components reference

This section contains information on the use of steel components that ship with Tekla Structures.

If you know which component you need, you can press F1 in the component dialog box to quickly access the correct help page. Some components use locally installed help files in a legacy format, which you can only access by pressing F1 in the component dialog box.

More components are available in Tekla Warehouse for you to download and install.

You can also modify many of the existing components and create your own custom components, see .

5.1 Shear tab connections

This section introduces components that can used in shear tab connections.

- Welded shear plate (43) (page 865)
- Shear plate (103) (page 881)
- Two sided shear plate (118) (page 894)
- Column with shear plate (131) (page 905)
- Bolted moment connection (134) (page 928)
- Shear plate simple (146) (page 954)
- Welded to top flange (147) (page 999)
- Welded to top flange S (149) (page 1025)
- Moment connection (181) (page 1052)
- Full depth (184) (page 1079)
- Full depth S (185) (page 1106)

- JP Full depth special (185) (page 1137)
- Shear plate tube column (189) (page 1152)

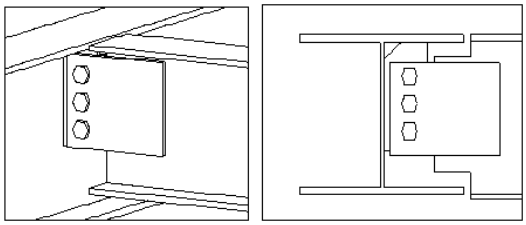
Welded shear plate (43)

Welded shear plate (43) connects two beams using a shear tab welded to the secondary beam web and bolted to a stiffener plate welded to the main part web.

Objects created

- Shear tab
- Stiffener
- Shim plate
- Bolts
- Welds

Use for

Situation	Description
	Shear tab welded to the secondary beam and bolted to a stiffener welded to the main part.

Limitations

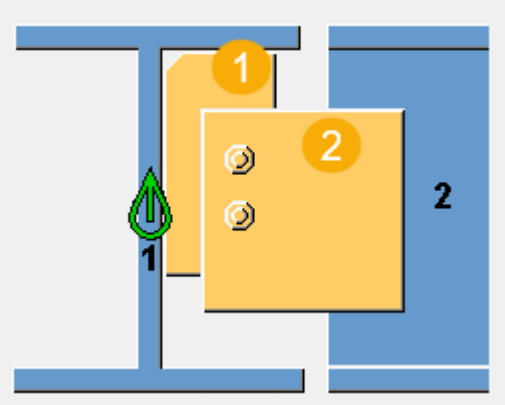
This connection cannot be used for connections between beams and columns.

Selection order

1. Select the main part (beam).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary beam is selected.

Part identification key

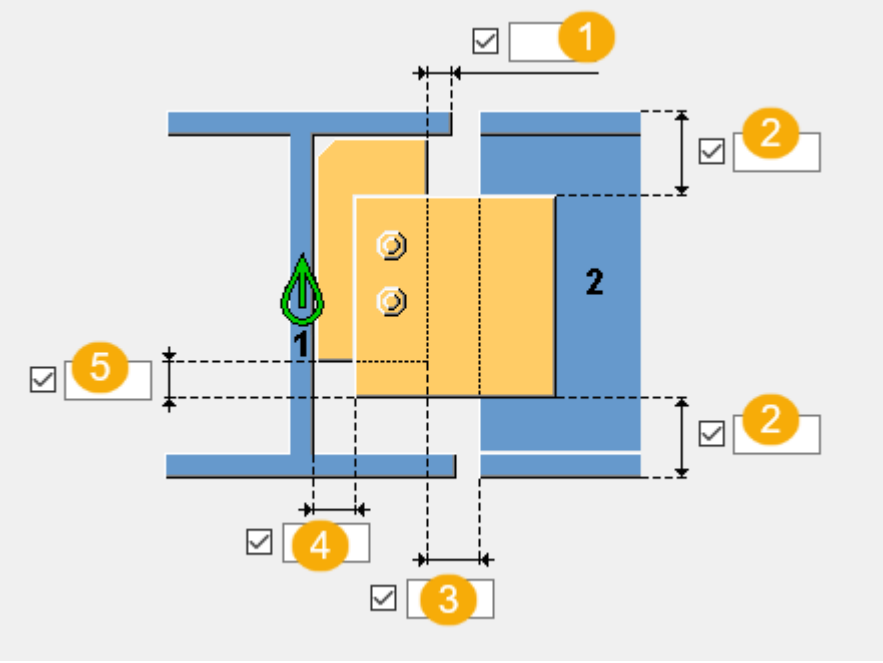


	Description
1	Stiffener
2	Shear tab

Picture tab

Use the **Picture** tab to define the plate positions.

Dimensions



	Description	Default
1	Edge distance between the stiffener and the main part flange.	0

	Description	Default
2	Shear tab top and bottom edge distance.	By default, the shear tab position and dimensions are defined by the bolt group edge distances. The default position for the shear tab top and bottom edges can be overwritten with the top and bottom edge distances.
3	Distance between the stiffener edge and secondary part end.	
4	Gap between the main part web and the shear tab.	`10 mm` Changing the value moves the bolt group correspondingly.
5	Edge distance between the shear tab and stiffener bottom.	

Parts tab

Use the **Parts** tab to define the part properties.

Parts

Option	Description	Default
Stiffener	Thickness, width and height of the stiffener.	By default, the height is defined by the bolt group vertical edge distances. If you do not enter a width, the stiffener width will be defined based on the flange width. The default value for the stiffener thickness is the web thickness.

Option	Description	Default
Plate	Thickness and width of the shear tab. The height is defined by the bolt group edge distances. You can also modify the height with shear tab top and bottom distances on the **Picture** tab.	The default value for the end plate thickness is half of the screw diameter.
Filler	Thickness of the shim plate.	

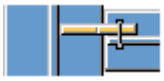
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

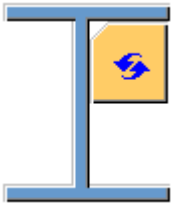
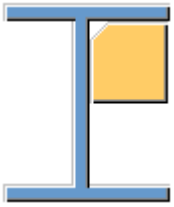
Use the **Parameters** tab to define the shear tab position, stiffener type and chamfers, and shim plate assembly.

Sheart tab position

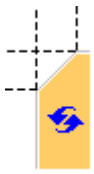
Option	Description
	Default Near side AutoDefaults can change this option.
	Automatic Near side

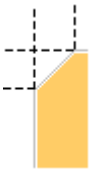
Option	Description
	Near side
	Both sides
	Far side

Stiffener creation

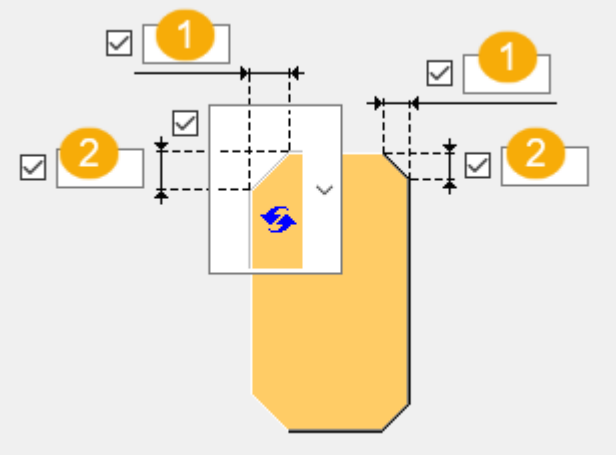
Option	Description
	Default Determined by shear tab AutoDefaults can change this option.
	Full
	Determined by shear tab Tekla Structures determines the size of the stiffener based on the shear tab size. Tekla Structures attempts to keep the bottom edges of the stiffener plate and shear tab level, if possible.

Chamfer type

Option	Description
	Default. Line chamfer AutoDefaults can change this option.

Option	Description
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Chamfer dimensions

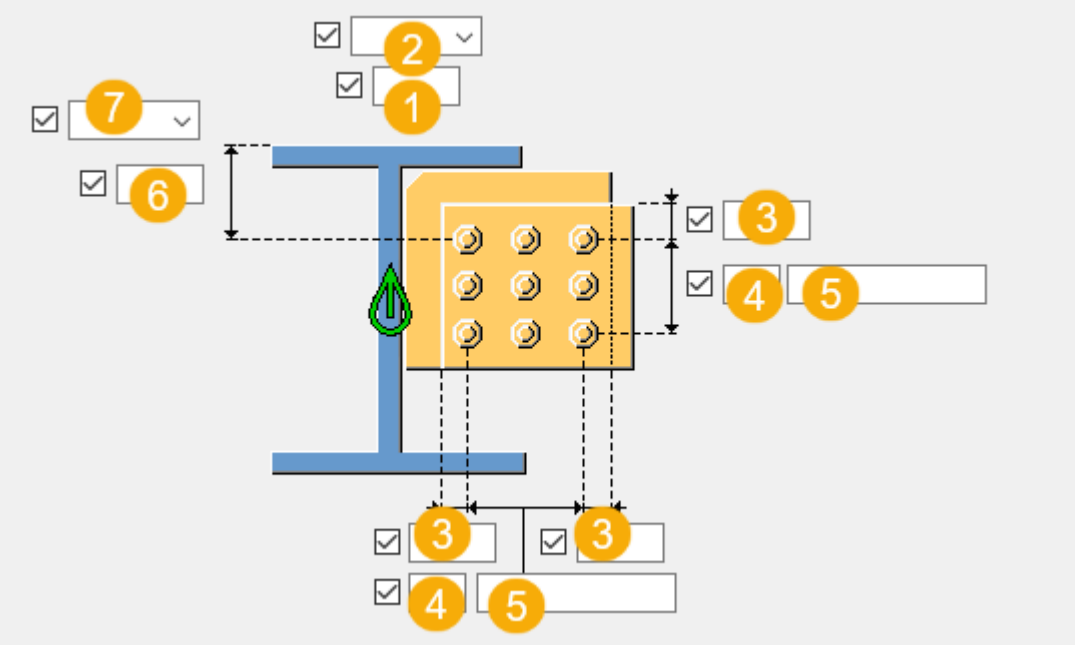


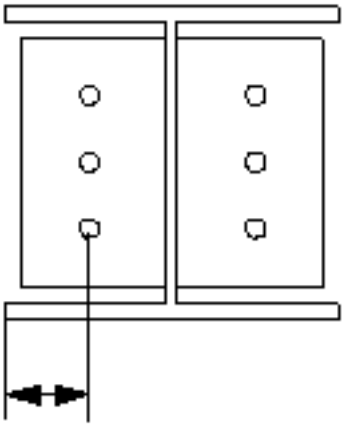
1	Horizontal chamfer dimension
2	Vertical chamfer dimension

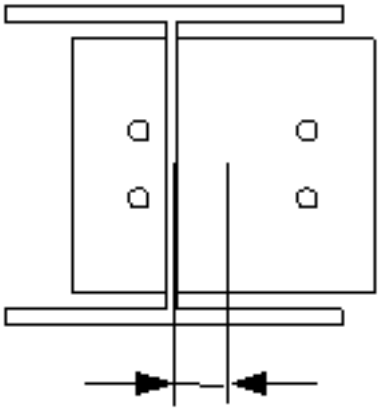
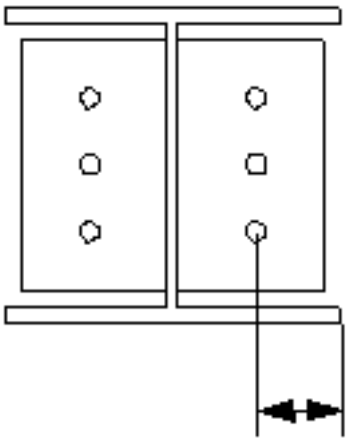
Bolts tab

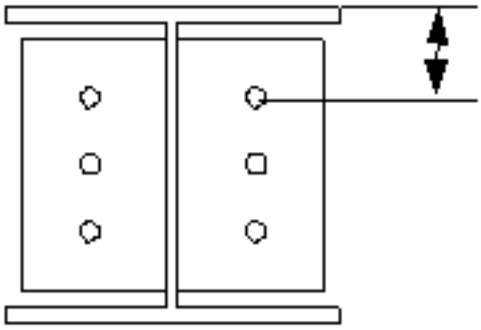
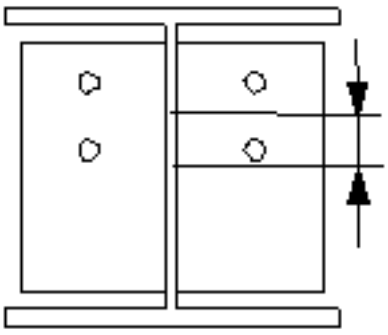
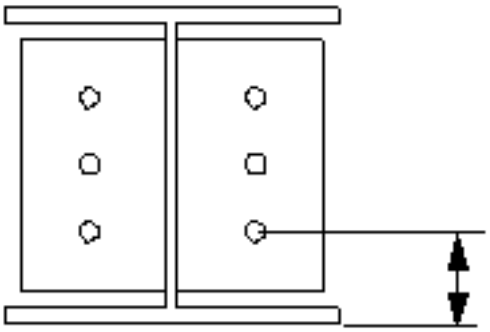
Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.
2	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt. 

	Description
	• **Middle**: From the center line of the secondary part to the center line of the bolts.  • **Right**: From the right edge of the secondary part to the rightmost bolt. 
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Dimension for vertical bolt group position.

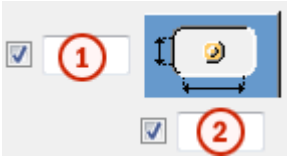
	Description
7	• **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt.  Select how to measure the dimensions for vertical bolt group position.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



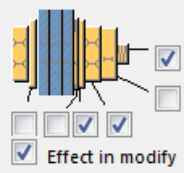
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	

Option	Description	Default
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beams and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

Automatic notching options affect both the top and the bottom flange.

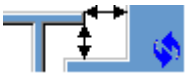
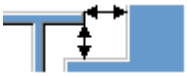
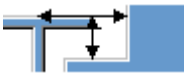
Notch shape

Automatic notching is switched on when you select a notch shape.

Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.

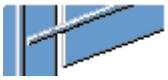
Option	Description
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

Enter the horizontal and vertical values for the cuts.

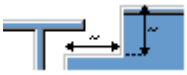
Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.

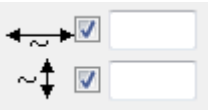
Option	Description
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.



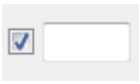
Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.



Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

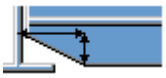
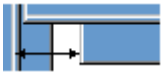
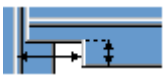
Side of flange notch

The side of flange notch defines on which side of the beam the notches are created.

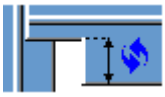
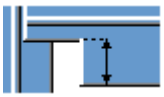
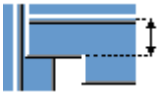
Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

Flange notch shape

The flange notch shape defines the notch shape in the beam flange.

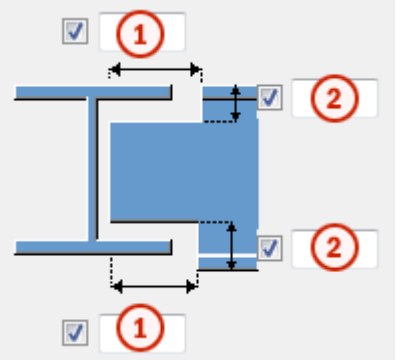
Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.

Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	`10 mm`
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest `5 mm`.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

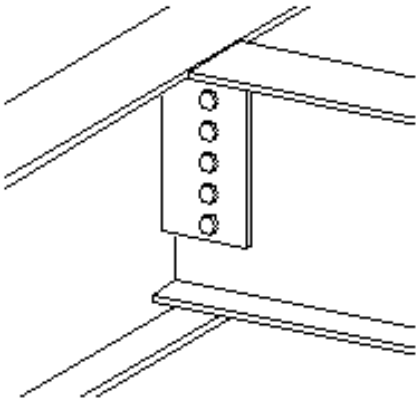
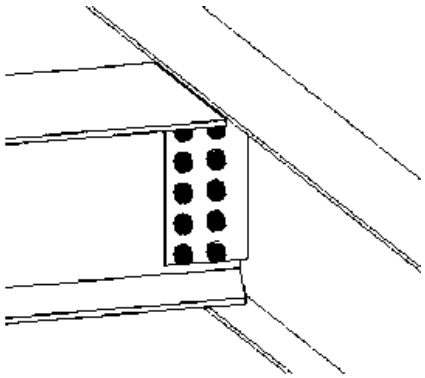
Shear plate (103)

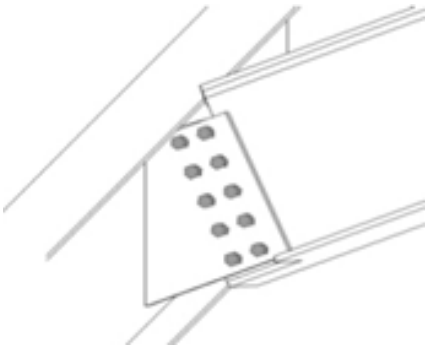
Shear plate (103) connects a beam to a beam or to a column with a shear tab. The shear tab is welded to the main part and bolted to the secondary beam.

Objects created

- Shear tab
- Bolts
- Welds
- Cuts

Use for

Situation	Description
	Shear tab connected to a beam.
	Shear tab connected to a beam. The secondary beam is skewed.

Situation	Description
	Shear tab connected to a beam. The secondary beam is sloped and skewed.

Selection order

1. Select the main part (beam or column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary beam is selected.

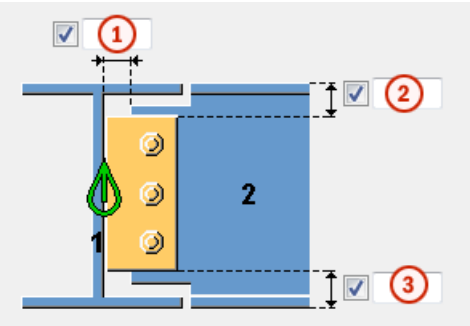
Part identification key

	Part
1	Shear tab

Picture tab

Use the **Picture** tab to control the position of the shear tab.

Shear tab dimensions

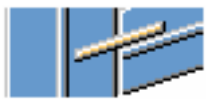


	Description
1	Cut of the secondary part. Cutting the secondary part creates a gap between the main part and the secondary part. The cut is defined from the main part web.
2	Distance from the top edge of the secondary beam to the top edge of the shear plate.
3	Distance from the bottom edge of the secondary beam to the bottom edge of the shear tab.

Shear tab position

Option	Description
	Default The shear tab is on the left side of the secondary beam web. AutoDefaults can change this option.
	The shear tab is on the left side of the secondary beam web.
	The shear tab is on the right side of the secondary beam web.

Beam flange cut

Option	Description
	Default Square AutoDefaults can change this option.
	Cuts the end of the flange bevel.
	Cuts the end of the flange square.

Parts tab

Use the **Parts** tab to control the shear tab properties.

Shear plate

Part	Description
Plate	Shear tab thickness and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

Automatic notching options affect both the top and the bottom flange.

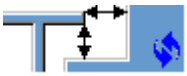
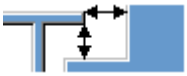
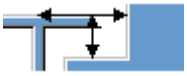
Notch shape

Automatic notching is switched on when you select a notch shape.

Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.

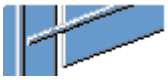
Option	Description
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

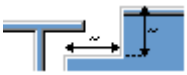
Enter the horizontal and vertical values for the cuts.

Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.

Notch position

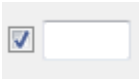
Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.

Option	Description
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.



Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

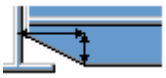
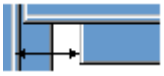
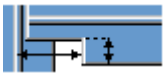
Side of flange notch

The side of flange notch defines on which side of the beam the notches are created.

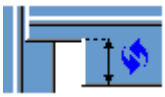
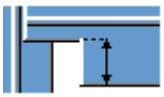
Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

Flange notch shape

The flange notch shape defines the notch shape in the beam flange.

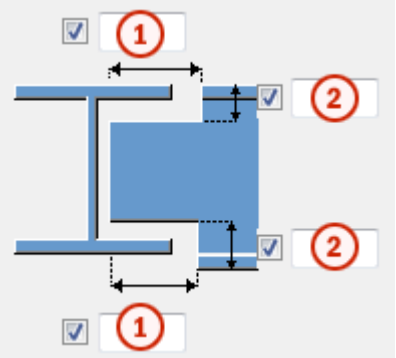
Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.

Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	10 mm
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest 5 mm.

BCSA notch definition

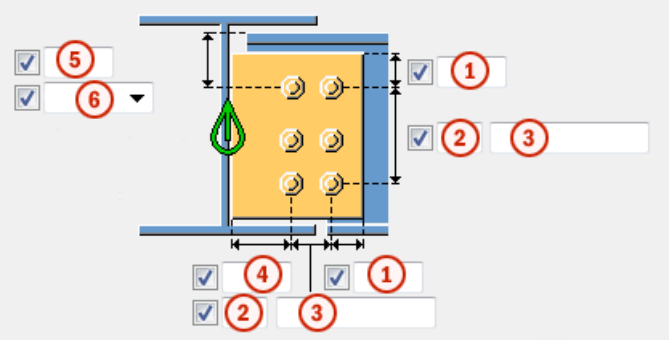
Define whether the notch is created according to British Constructional Steelwork Association (BCSA) specifications.

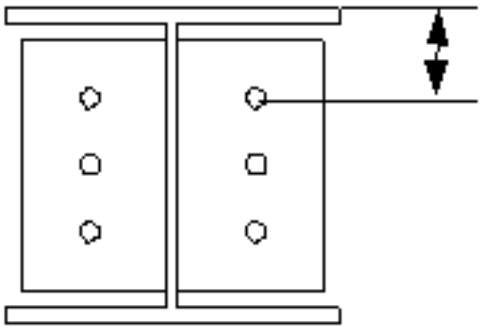
Option	Description
Default	Notch dimensions.
Yes	Creates a 50 mm notch for simple beam-to-beam connections.
No	Use the options on this **Notch** tab to define the notch dimensions.

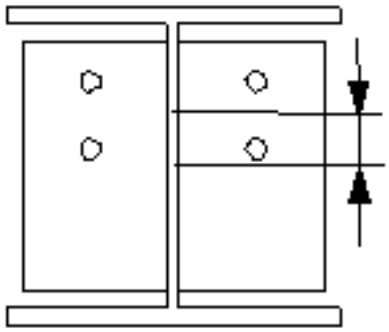
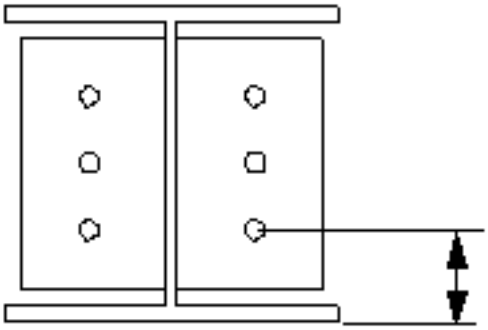
Bolts tab

Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions



	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Dimension for horizontal bolt group position.
5	Dimension for vertical bolt group position.
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3

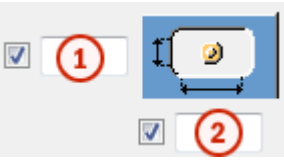
Option	Description
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



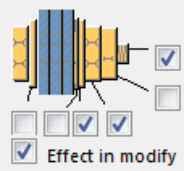
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Bolting direction

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1

Option	Description
	Bolting direction 2

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

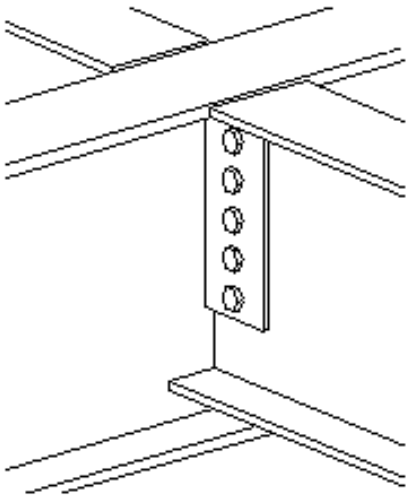
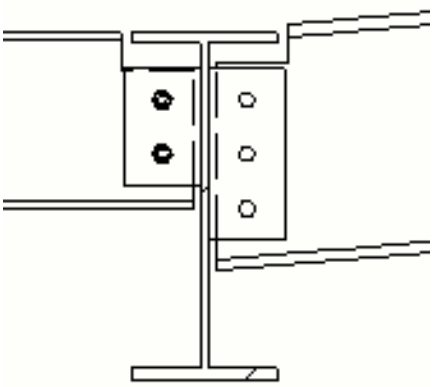
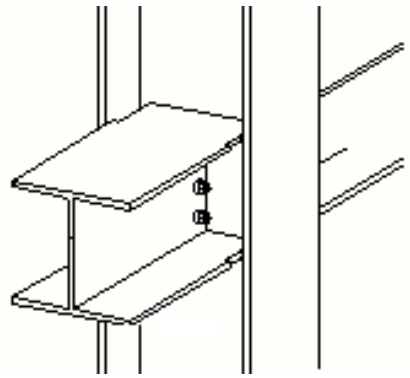
Two sided shear plate (118)

Two sided shear plate (118) connects two beams to a beam or a column with shear tabs. The shear tabs are welded to the main beam or column, and bolted to the secondary beams.

Objects created

- Shear tabs (2)
- Bolts
- Welds
- Cuts

Use for

Situation	Description
	Simple shear tabs connected to a beam.
	Simple shear tabs connected to a beam. The other secondary beam is sloped.
	Simple shear tabs connected to a column web.

Limitations

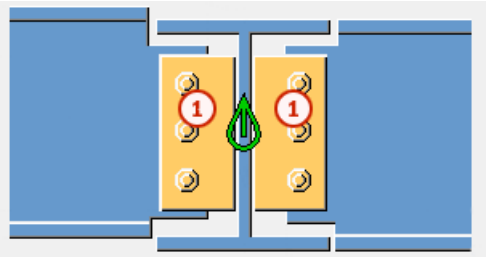
The upper edges on the shear tabs should be on the same level.

Selection order

1. Select the main part (beam or column).

2. Select the first secondary part (beam).
3. Select the second secondary part (beam).
4. Click the middle mouse button to create the connection.

Part identification key

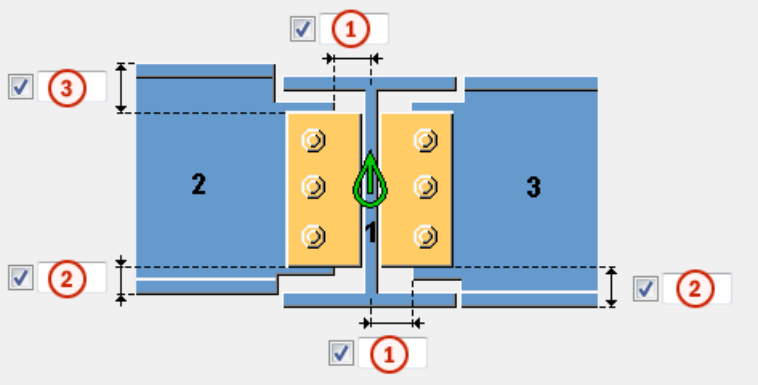


	Part
1	Shear tab

Picture tab

Use the **Picture** tab to control the position of the shear tabs.

Shear tab dimensions



	Description	Default
1	Cut of the secondary part. Cutting the secondary part creates a gap between the main part and the secondary part. The cut is defined from the main part web.	
2	Distance from the bottom edge of the secondary beam to the bottom edge of the shear tab.	

	Description	Default
3	Distance from the top edge of the first secondary beam to the top edge of the shear plate. The top edges of the shear tabs are aligned to the same level.	50 mm

Shear tab position

Option	Description
	Default The shear tab is on the left side of the secondary beam web. AutoDefaults can change this option.
	The shear tab is on the left side of the secondary beam web.
	The shear tab is on the right side of the secondary beam web.

Beam flange cut

Option	Description
	Default Square AutoDefaults can change this option.
	Bevel Cuts the end of the flange bevel.
	Square Cuts the end of the flange square.

Parts tab

Use the **Parts** tab to control the shear tab properties.

Part	Description
Plate	Shear tab thickness and height.

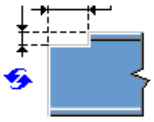
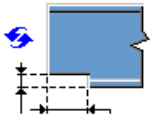
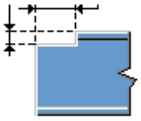
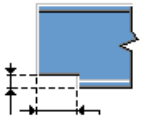
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

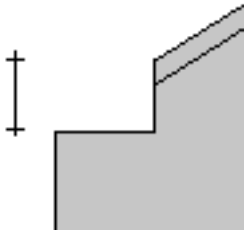
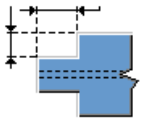
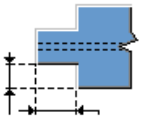
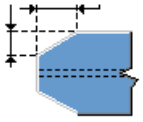
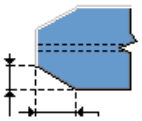
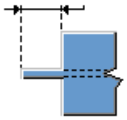
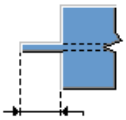
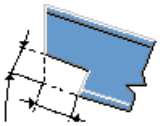
Notch tab

Use the **Notch** tab to create notches for the secondary beams and to control the notch properties. Define the notches for both secondary beams.

Notch shape

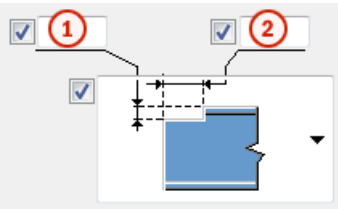
Define the notch shape for the top and the bottom of the secondary beam.

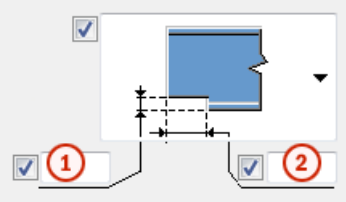
Option	Option	Description
		Default Creates a square notch on the top side or on the bottom side of the secondary beam. AutoDefaults can change this option.
		No notch
		Creates a square notch on the top side or on the bottom side of the secondary beam. Define the notch dimensions. In beam-to-beam connections with a

Option	Option	Description
		sloped secondary beam, the depth is measured as shown in the picture. 
		Creates a notch on both sides of the secondary part. Define the notch dimensions.
		Creates a chamfered notch on both sides of the secondary beam. Define the chamfer dimensions.
		Creates a strip. Define the length of the strip. The flanges are cut completely.
		Creates a special type of square notch. Define the notch dimensions. The notch is square to the secondary beam. There are no default values for the length or the depth.

Notch dimensions

Define the top and the bottom dimensions of the notch if you have set the **BCSA notch def** option to **No**.





	Description
1	Vertical notch dimension.
2	Horizontal notch dimension.

Notching side

Define on which side of the secondary beam the notch is created. You can define the side for both the top and the bottom of the secondary beam.

Option	Description
	Default Creates notches on both sides. AutoDefaults can change this option.
	Creates notches on both sides.
	Creates a notch on the left side.
	Creates a notch on the right side.

BCSA notch definition

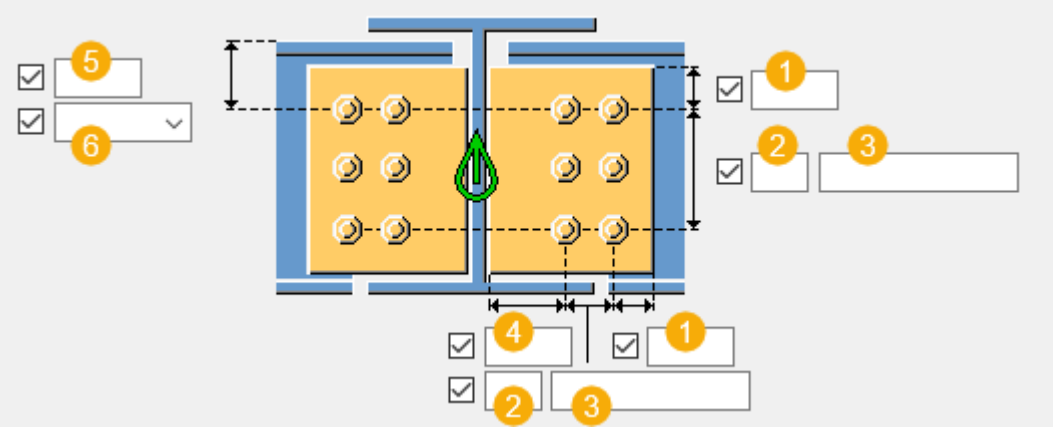
Define whether the notch is created according to British Constructional Steelwork Association (BCSA) specifications.

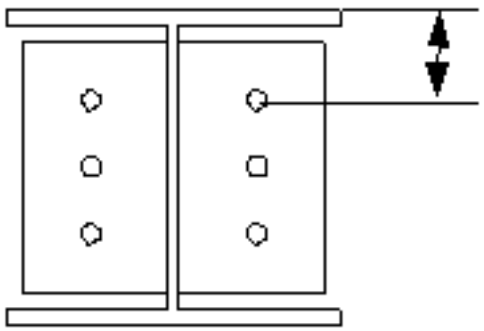
Option	Description
Default	Notch dimensions.
Yes	Creates a 50 mm notch for simple beam-to-beam connections.
No	Use the options on this **Notch** tab to define the notch dimensions.

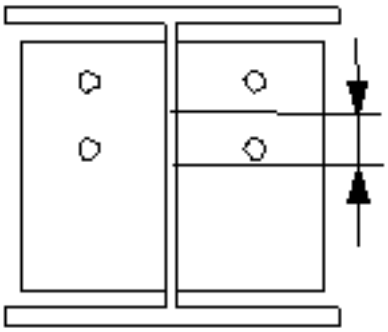
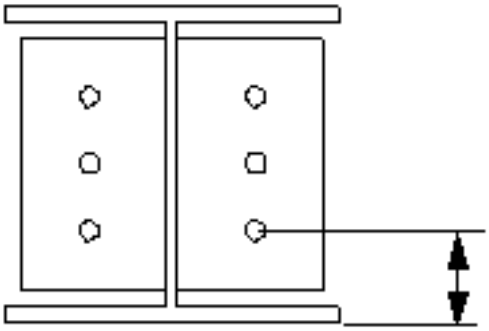
Bolts tab

Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions



	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Define the horizontal bolt edge distance.
5	Dimension for vertical bolt group position.
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3

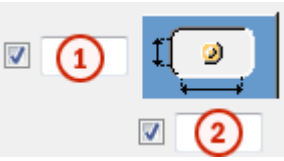
Option	Description
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



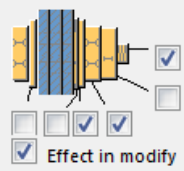
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

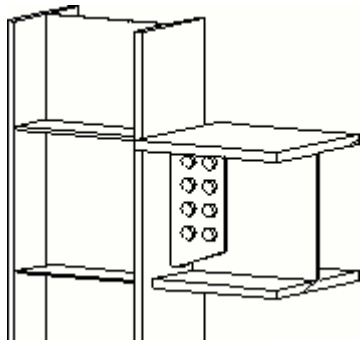
Column with shear plate (131)

Column with shear plate (131) connects a beam to a column with a single shear tab or double shear tabs. The shear tab is welded to the main part web and stiffeners, and bolted to the secondary part web. The secondary beam can be leveled or sloped.

Objects created

- Shear tabs (1 or 2)
- Stiffeners (optional)
- Welds
- Bolts
- Cuts

Use for

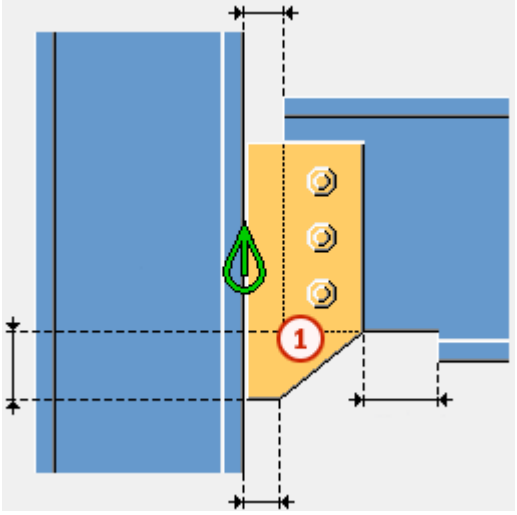
Option	Description
	Two shear tabs and four stiffeners.

Selection order

1. Select the main part (column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key



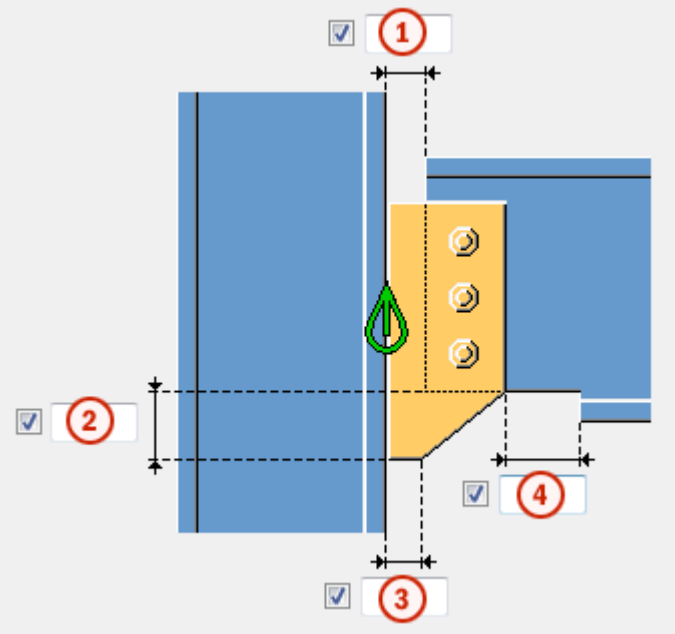
	Part
1	Shear tab

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to control the position of the shear tab, and the beam flange and the web cuts.

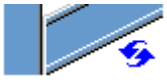
Dimensions

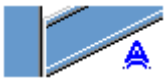


	Description	Default
1	Cut of the secondary part. Cutting the secondary part creates a gap between the main part and the secondary part.	10 mm
2	Height of the chamfered part of the shear tab.	50 mm
3	Distance from the edge of the main part to the corner of the shear tab.	20 mm
4	Size of the strip made to the secondary part flange. The cut of the flange is defined from the shear tab edge.	20 mm

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.

Option	Description
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.
	Square Cuts the end of the secondary beam square.
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.

Beam web cut

Define how the secondary beam web end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the web bevel when the end of the secondary beam is cut bevel.
	Square Cuts the end of the web square even if the end of the secondary beam is cut bevel.

Beam flange cut

Define how the secondary beam flange end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the flange bevel.

Option	Description
	Square Cuts a part of the flange square and leaves a part of it bevel.

Beam bottom flange cut

Option	Description
	Default Notch Define the notch dimensions. AutoDefaults can change this option.
	Notch Define the notch dimensions. The bottom of the secondary beam is notched if the shear tab crosses the flange.
	Flange cut The secondary beam flange is cut on the same side as the shear tab if the shear tab crosses the flange.

Plates tab

Use the **Plates** tab to control the size, position, number, orientation and shape of the shear tab.

Shear tab plate

Option	Description
Tab plate	Shear tab plate thickness, width and height.

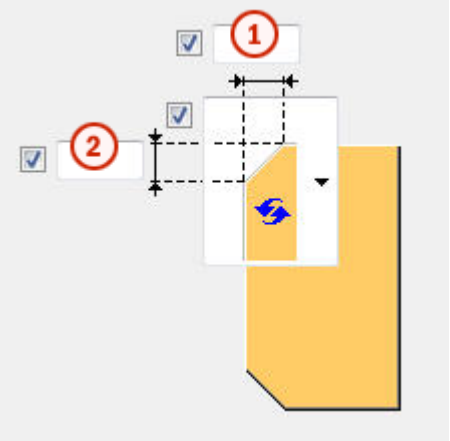
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Shear tab shape

Option	Description
	Default Shear tab corner is chamfered. AutoDefaults can change this option.
	Automatic Shear tab corner is chamfered.
	Shear tab corner is chamfered.
	Shear tab corner is not chamfered.

Shear tab chamfers



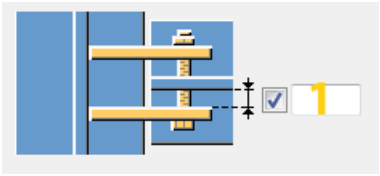
	Description
1	Horizontal dimension of the shear tab chamfer.
2	Vertical dimension of the shear tab chamfer.

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer

Option	Description
	Concave arc chamfer

Gap between shear tabs



	Description	Default
1	Gap between the secondary part web and shear tab. This only affects connections with two shear tabs.	0

Shear tab position

Define the number and the side of shear tabs in single shear tab connections.

Option	Description
	Default Far side shear tab AutoDefaults can change this option.
	Automatic The component automatically selects either the near side or the far side shear tab. The tab is created to the side of the secondary part when the angle between the main part and the secondary part is less than 90 degrees.
	Far side shear tab

Option	Description
	Near side and far side shear tab
	Near side shear tab

Shear tab orientation

Option	Description
	Default Sloped AutoDefaults can change this option.
	Automatic The shear tab is sloped in the direction of the secondary beam. Both vertical edges of the shear tab are cut parallel to the end of the secondary beam.
	Sloped The shear tab is sloped in the direction of the secondary beam. Both vertical edges of the shear tab are cut parallel to the end of the secondary beam.
	Square
	Modified sloped Same as the **Sloped** option, but the vertical edge of the shear tab connected to the secondary beam is cut perpendicular to the secondary beam flange.

Stiffeners tab

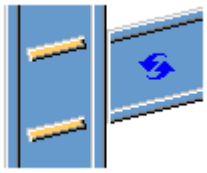
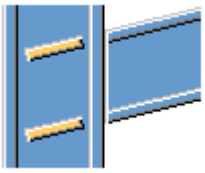
Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position and type.

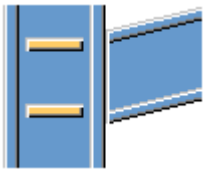
Stiffener plate dimensions

Option	Description
Top NS	Top near side stiffener thickness, width and height.
Top FS	Top far side stiffener thickness, width and height.
Bottom NS	Bottom near side stiffener thickness, width and height.
Bottom FS	Bottom far side stiffener thickness, width and height.

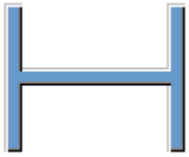
Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Stiffener orientation

Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are parallel to the secondary part.

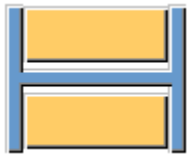
Option	Description
	Stiffeners are perpendicular to the main part.

Stiffener creation

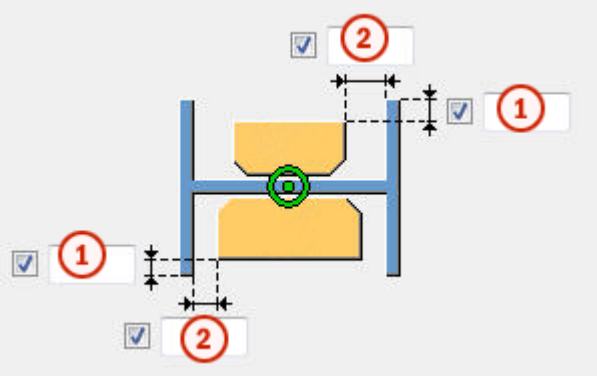
Option	Description
	Default Stiffeners are created. AutoDefaults can change this option.
	Automatic Stiffeners are created when necessary.
	No stiffeners are created.
	Stiffeners are created.

Stiffener shape

Option	Description
	Default Line chamfered stiffener plates AutoDefaults can change this option.
	Automatic Line chamfered stiffener plates

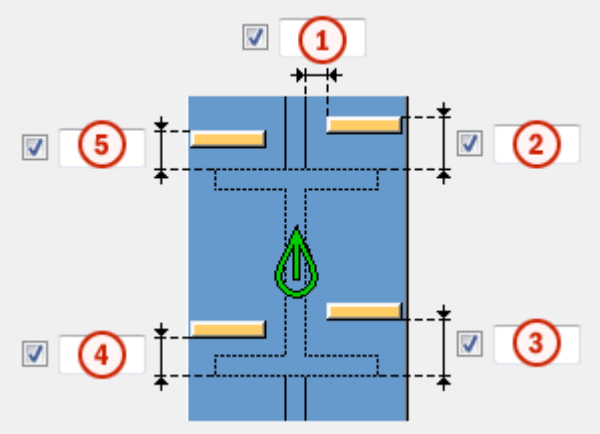
Option	Description
	Square stiffener plates Stiffener plates with a gap for the main part web rounding
	Line chamfered stiffener plates

Stiffener gap



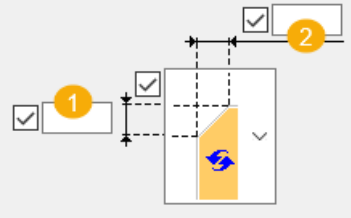
	Description
1	Distance from the edge of the flange to the edge of the stiffener.
2	Size of the gap between the flanges and the stiffener.

Stiffener positions



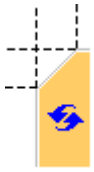
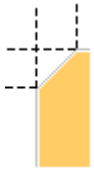
	Description
1	Size of the gap between the stiffener and the beam web edge.
2	Size of the gap between the top near side stiffener and the beam flange edge.
3	Size of the gap between the bottom near side stiffener and the beam flange edge.
4	Size of the gap between the bottom far side stiffener and the beam flange edge.
5	Size of the gap between the top far side stiffener and the beam flange edge.

Chamfer dimensions



	Description	Default
1	Vertical dimension of the chamfer.	`10 mm`
2	Horizontal dimension of the chamfer.	`10 mm`

Chamfer type

Option	Description
	Default. Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer

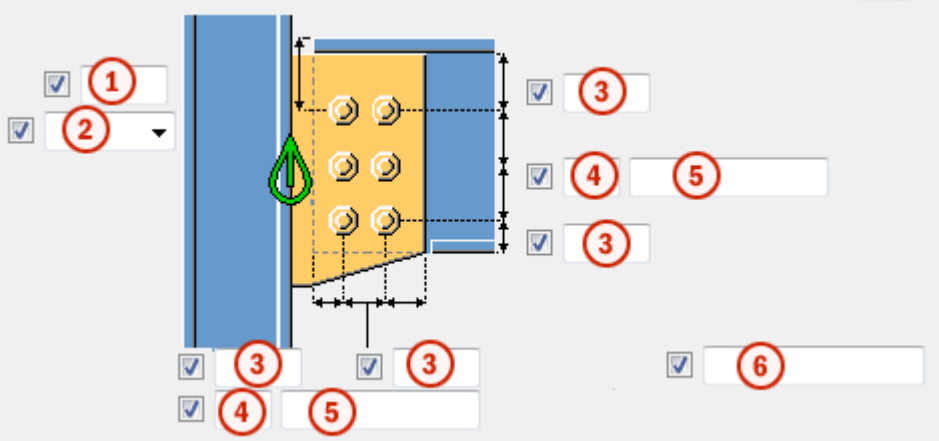
Option	Description
	Convex arc chamfer
	Concave arc chamfer

Bolts tab

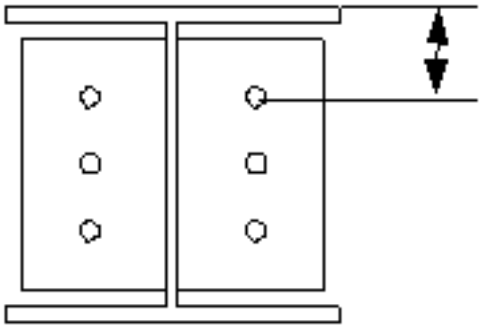
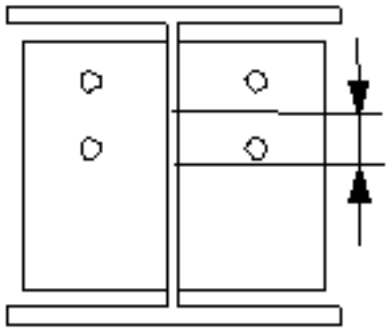
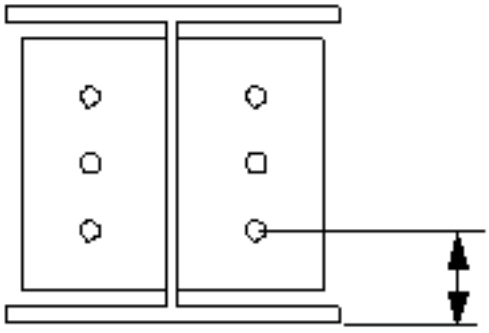
Use the **Bolts** tab to control the properties of the bolts that connect the shear tab to the secondary part.

Bolt group dimensions

Bolt group dimensions affect the size and shape of the shear tab.



	Description
1	Dimension for vertical bolt group position.

	Description
2	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

	Description
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt group orientation

Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square

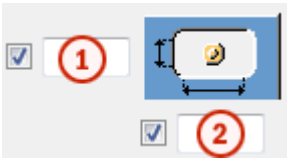
Option	Description
	Staggered Bolts are staggered in the direction of the secondary part.
	Square Square bolt group is positioned horizontally.
	Sloped Square bolt group is sloped in the direction of the secondary part.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



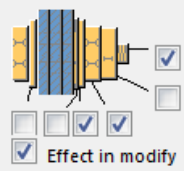
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.

Option	Description	Default
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

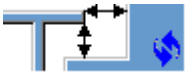
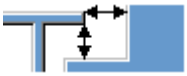
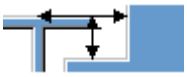
Automatic notching options affect both the top and the bottom flange.

Notch shape

Automatic notching is switched on when you select a notch shape.

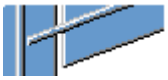
Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

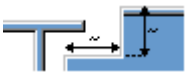
Enter the horizontal and vertical values for the cuts.

Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.

Notch position

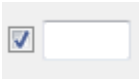
Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.

Option	Description
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.



Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

Side of flange notch

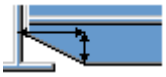
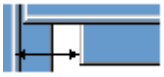
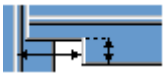
The side of flange notch defines on which side of the beam the notches are created.

Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.

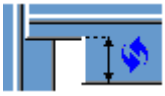
Option	Description
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

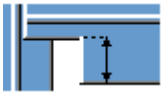
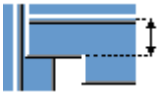
Flange notch shape

The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

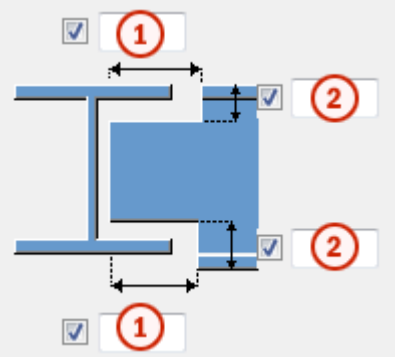
Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.

Option	Description
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.

Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	`10 mm`
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest `5 mm`.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design type tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

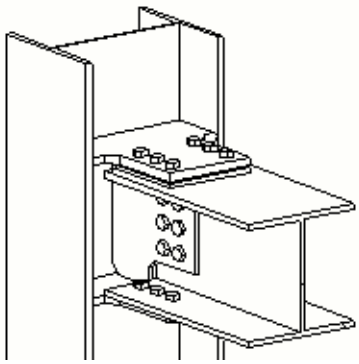
Bolted moment connection (134)

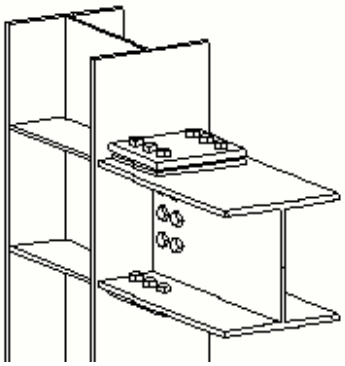
Bolted moment connection (134) connects a beam to a column web or flange. The shear tab is welded to the main part web or flange, and bolted to the secondary part web. The secondary part can be leveled or sloped and/or skewed.

Objects created

- Shear tab (1 or 2)
- Flange plates (2)
- Shim plates
- Stiffeners (optional)
- Web doubler plate (optional)
- Bolts
- Welds
- Cuts

Use for

Situation	Description
	Beam connected to column web.

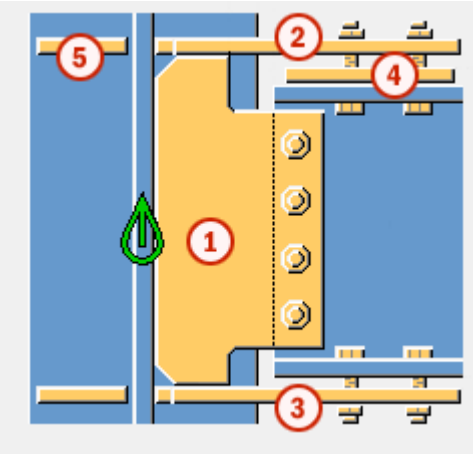
Situation	Description
	Beam connected to column flange.

Selection order

1. Select the main part (column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key



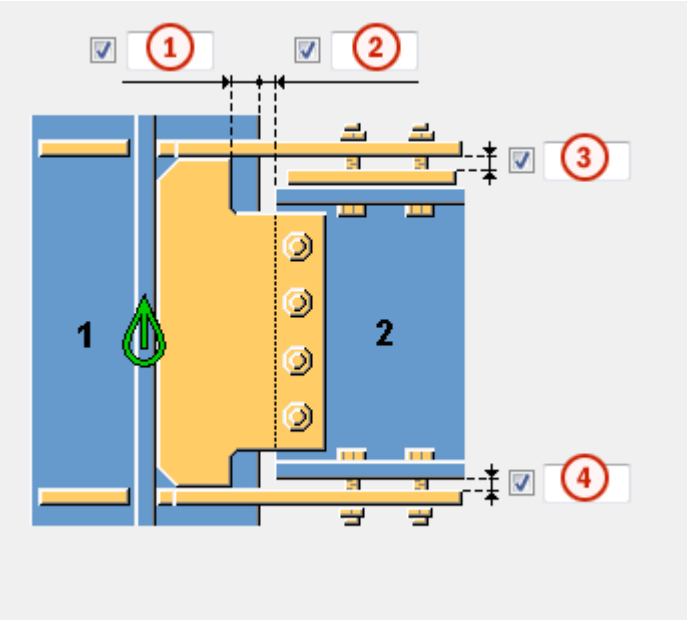
	Part
1	Shear tab
2	Upper flange plate
3	Lower flange plate
4	Shim plate
4	Stiffener

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to control the position of the shear tab, and the beam flange and web cuts.

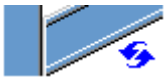
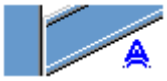
Dimensions



	Description
1	Shear tab edge distance from the main part flange edge.
2	Cut of the secondary part. Cutting the secondary part creates a gap between the main part and the secondary part.
3	Distance from the shim plate edge to the flange plate edge.
4	Distance from the edge of the secondary part to the edge of the flange plate.

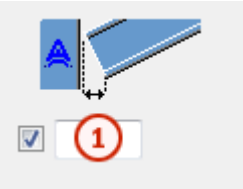
Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut

Option	Description
	square. Otherwise, the beam end is cut bevel.
	Square Cuts the end of the secondary beam square.
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.

Beam bevel cut



	Description
1	Bevel of the beam end cut.

Beam flange cut

Define how the secondary beam flange end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the flange bevel.
	Square Cuts a part of the flange square and leaves a part of it bevel.

Shear plate tab

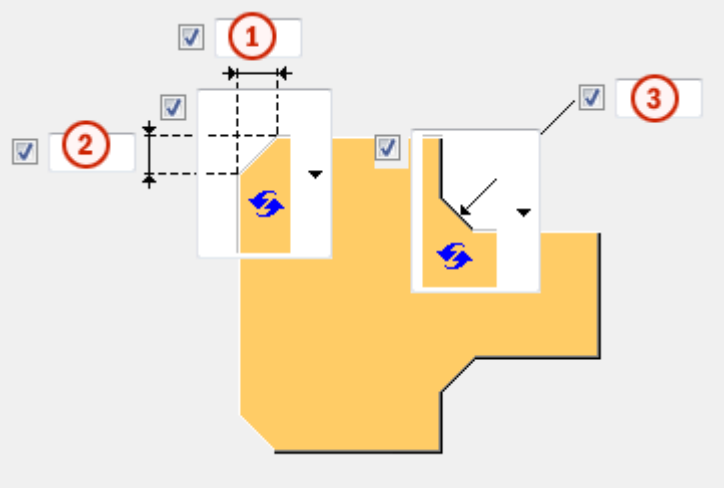
Use the **Shear plate** tab to control the size, position, number, orientation and shape of the shear tab.

Shear tab

Option	Description
Tab plate	Shear tab plate thickness, width and height.

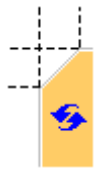
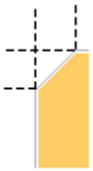
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Shear tab chamfers



	Description
1	Horizontal dimension of the shear tab chamfer.
2	Vertical dimension of the shear tab chamfer.
3	Vertical and the horizontal dimension of the shear tab chamfer.

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Chamfer type dimensions

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer

Option	Description
	Line chamfer
	Concave arc chamfer

Shear tab orientation

Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Sloped The shear tab is sloped in the direction of the secondary beam. Both vertical edges of the shear tab are cut parallel to the end of the secondary beam.
	Square

Shear tab position

Define the number and the side of shear tabs in single shear tab connections.

Option	Description
	Default Far side shear tab AutoDefaults can change this option.
	Automatic The component automatically selects either the near side or the far side shear tab. The tab is created to the side of the secondary part when the

Option	Description
	angle between the main part and the secondary part is less than 90 degrees.
	Far side shear tab
	Near side and far side shear tab
	Near side shear tab

Flange plate tab

Use the **Flange plate** tab to control the size, position, number, orientation and shape of the flange plates and shim plates.

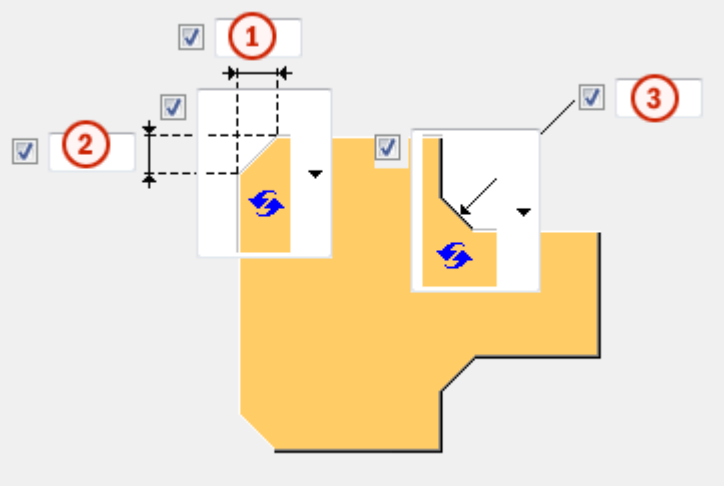
Plates

Part	Description	Default
Upper flange plate	Upper flange plate thickness.	`20 mm`
Lower flange plate	Lower flange plate thickness.	`20 mm`
Upper deck fill	Upper deck shim plate thickness and width.	`10 mm`
Lower deck fill	Lower deck shim plate thickness and width.	`0`
Upper loose fill	Upper shim plate thickness.	
Lower loose fill	Lower shim plate thickness.	

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

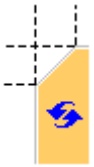
Option	Description	Default
	assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

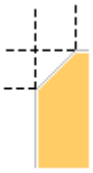
Flange plate chamfers



	Description
1	Horizontal dimension of the flange plate chamfer.
2	Vertical dimension of the flange plate chamfer.
3	Vertical and horizontal dimension of the flange plate chamfer.

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.

Option	Description
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Chamfer type dimensions

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer

Option	Description
	Concave arc chamfer

Stiffeners tab

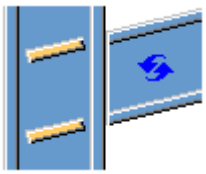
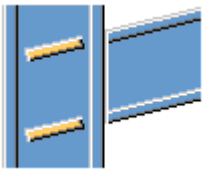
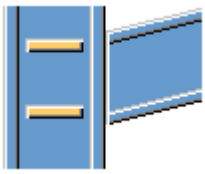
Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position, and type.

Stiffener plate dimensions

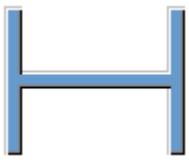
Option	Description
Top NS	Top near side stiffener thickness, width and height.
Top FS	Top far side stiffener thickness, width and height.
Bottom NS	Bottom near side stiffener thickness, width and height.
Bottom FS	Bottom far side stiffener thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Stiffener orientation

Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are parallel to the secondary part.
	Stiffeners are perpendicular to the main part.

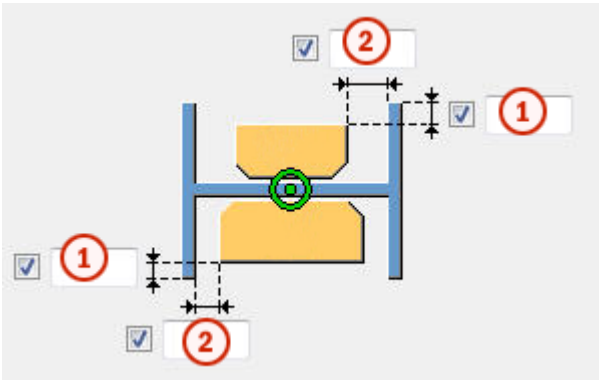
Stiffener creation

Option	Description
	Default Stiffeners are created. AutoDefaults can change this option.
	Automatic Stiffeners are created when necessary.
	No stiffeners are created.
	Stiffeners are created.

Stiffener shape

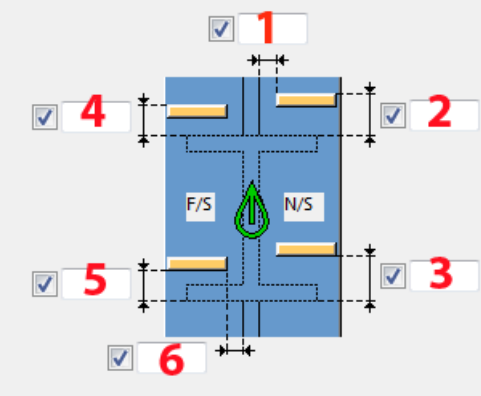
Option	Description
	Default Line chamfered stiffener plates AutoDefaults can change this option.
	Automatic Line chamfered stiffener plates
	Square stiffener plates Stiffener plates with a gap for the main part web rounding
	Line chamfered stiffener plates

Stiffener gap



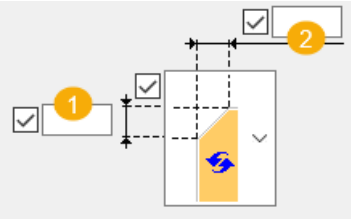
	Description
1	Distance from the edge of the flange to the edge of the stiffener.
2	Size of the gap between the flanges and the stiffener.

Stiffener positions



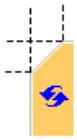
	Description
1	Size of the gap between the near side stiffener and the beam web edge.
2	Size of the gap between the top near side stiffener and the beam flange edge.
3	Size of the gap between the bottom near side stiffener and the beam flange edge.
4	Size of the gap between the top far side stiffener and the beam flange edge.
5	Size of the gap between the bottom far side stiffener and the beam flange edge.
6	Size of the gap between the far side stiffener and the beam web edge.

Chamfer dimensions



	Description	Default
1	Vertical dimension of the chamfer.	`10 mm`
2	Horizontal dimension of the chamfer.	`10 mm`

Chamfer type

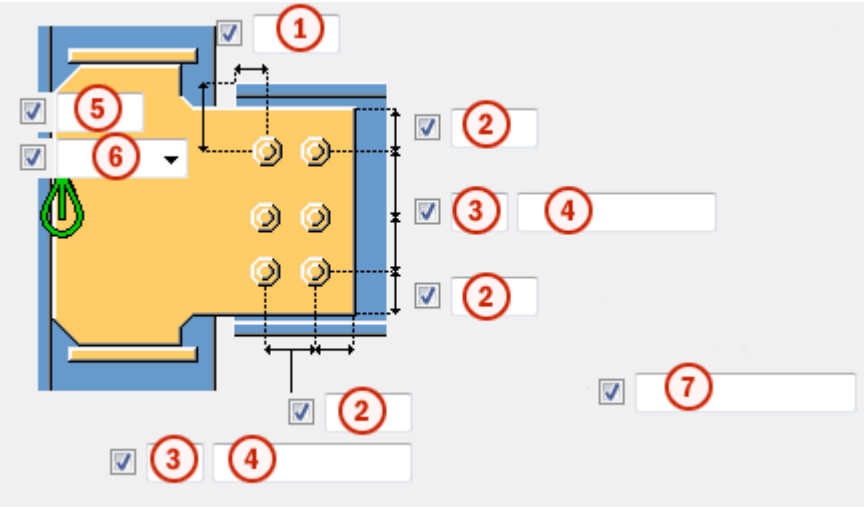
Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer
	Line and arc chamfer

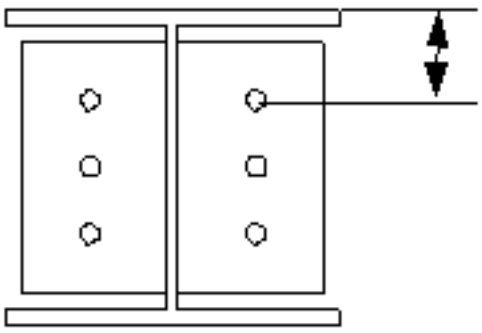
Shear bolts tab

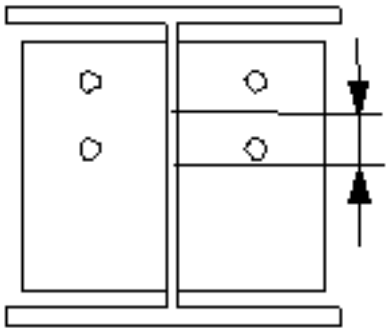
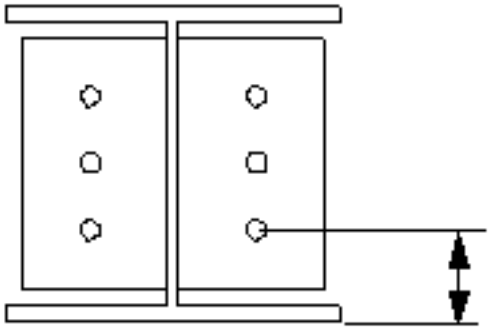
Use the **Shear bolts** tab to control the properties of the bolts that connect the shear tab to the secondary part.

Bolt group dimensions

Bolt group dimensions affect the size and shape of the shear tab.

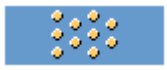


	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position.
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1

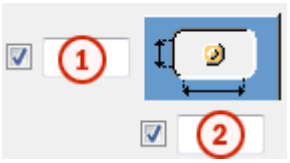
Option	Description
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



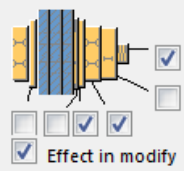
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.

Option	Description	Default
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

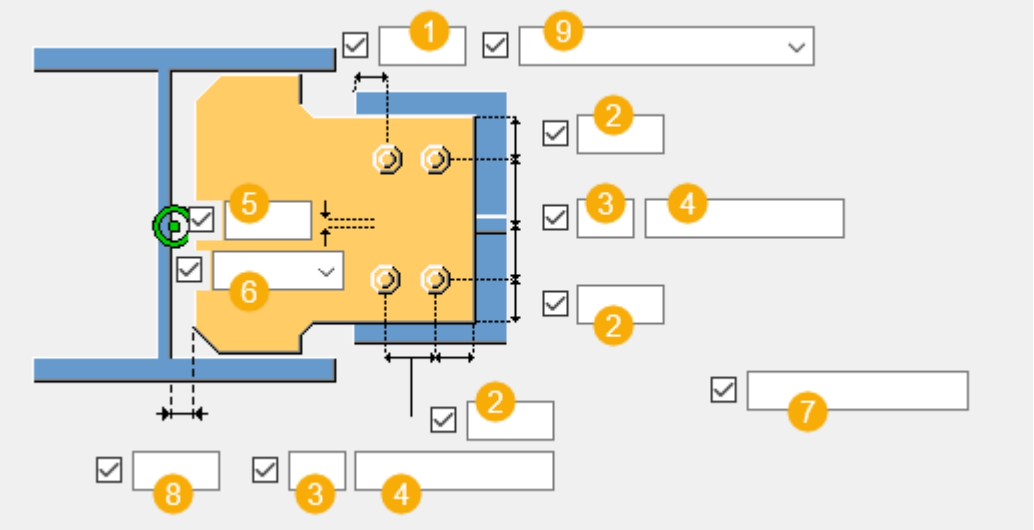
Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

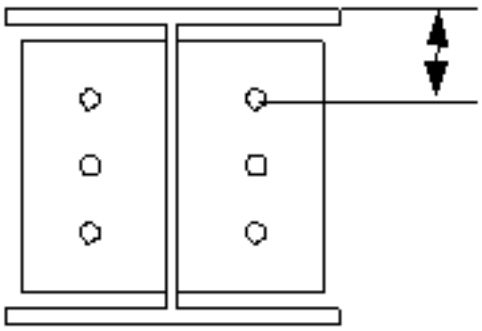
Flange bolts tab

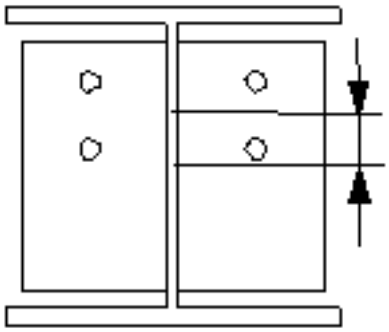
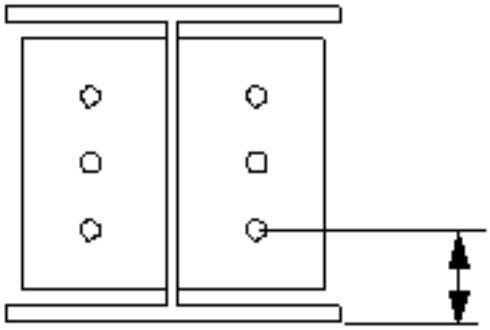
Use the **Flange bolts** tab to control the properties of the bolts that connect the flange plate to the secondary part.

Bolt group dimensions

Bolt group dimensions affect the size and shape of the flange plate.



	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position.
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.
8	Flange plate edge distance from the main part web.
9	Select the offset origin part for the bolt group.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered

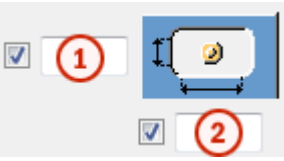
Option	Description
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.

Option	Description	Default
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Slots in deck fill plates and loose fill plates

Option	Description
Slots in deck fill plates, **Slots in loose fill plates**	Select whether slottes holes are created in deck fill and loose fill plates.

Doubler plate tab

Use the **Doubler plate** tab to create doubler plates to strengthen the web of the main part.

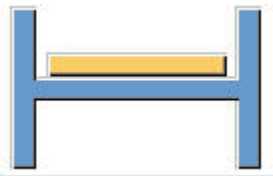
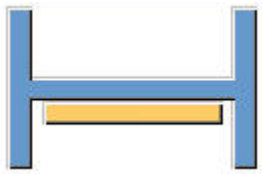
Web plate

Option	Description
Web plate	Web plate thickness and height.

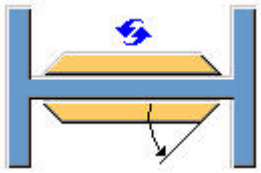
Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in

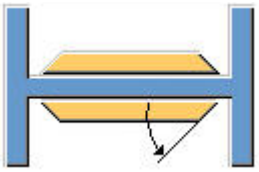
Option	Description	Default
		File menu --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Doubler plates

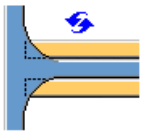
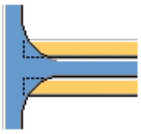
Option	Description
	Default Doubler plates are not created. AutoDefaults can change this option.
	Doubler plates are not created.
	Doubler plate is created on the far side.
	Doubler plate is created on the near side.
	Doubler plates are created on both sides.

Doubler plate edge shape

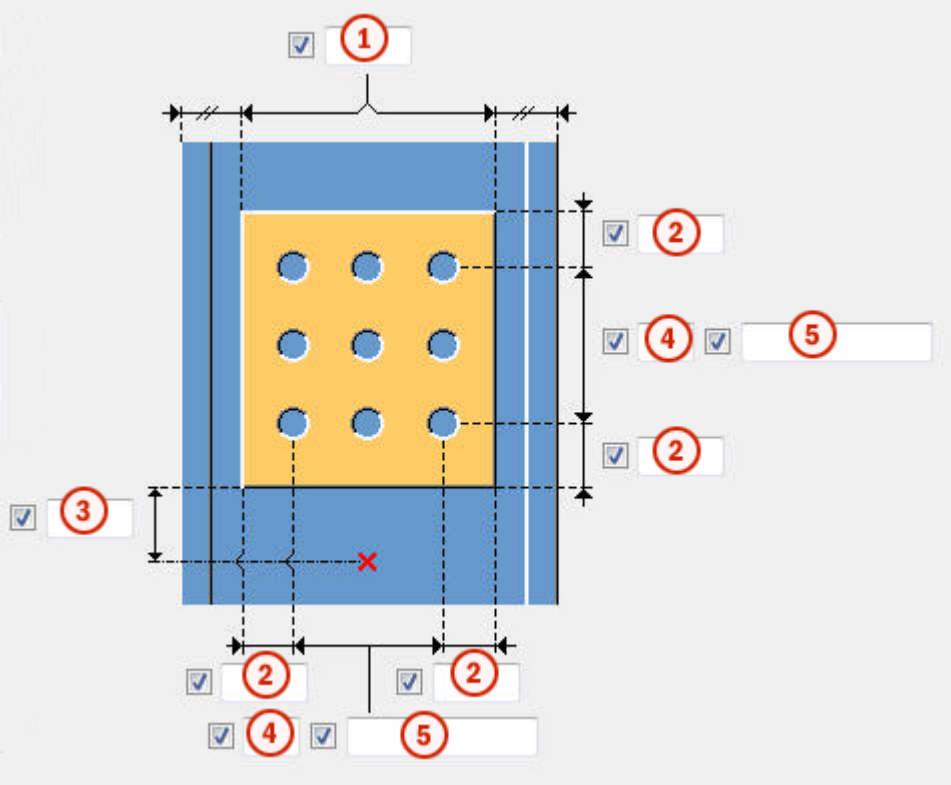
Option	Description
	Default Bevel doubler plates AutoDefaults can change this option.

Option	Description
	Bevel doubler plates Enter the angle in
	Square doubler plates

Doubler plate cuts

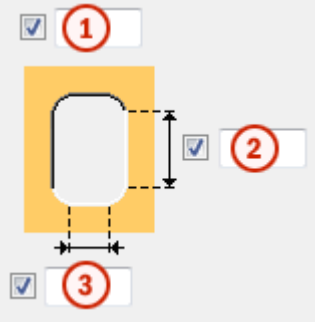
Option	Description
	Default Doubler plates are not cut. AutoDefaults can change this option.
	Doubler plates are not cut.
	Doubler plates are cut in the area that connects the main part web and flange.

General settings



	Description
1	Edge distance from the column flange.
2	Doubler plate edge distance. Edge distance is the distance from the center of a hole to the edge of the part.
3	Edge distance of the doubler plate in relation to the bottom of the secondary part.
4	Number of holes.
5	Hole spacing. Use a space to separate hole spacing values. Enter a value for each space between holes. For example, if there are 3 holes, enter 2 values.

Weld hole size



	Description
1	Hole diameter.
2	Slot length.
3	Slot width.

General tab

Click the link below to find out more:

General tab

Design type tab

Click the link below to find out more:

Design type tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Shear plate simple (146)

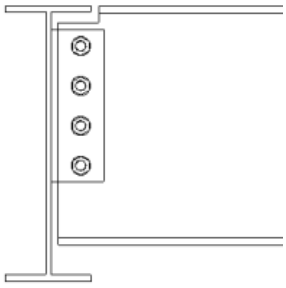
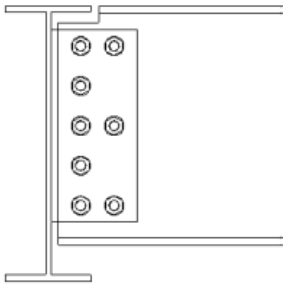
Shear plate simple (146) connects a beam to a beam, or a beam to a column with a single square shear tab or double shear tabs. The shear tab is welded to the main part web and flanges, and bolted to the secondary beam web. The

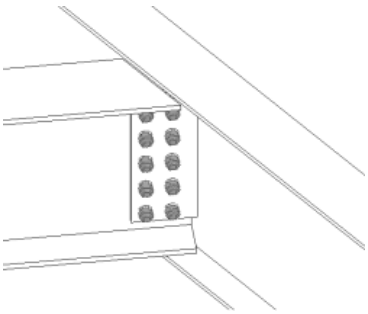
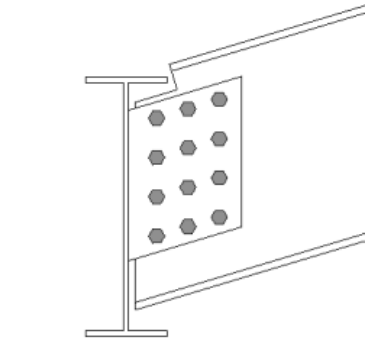
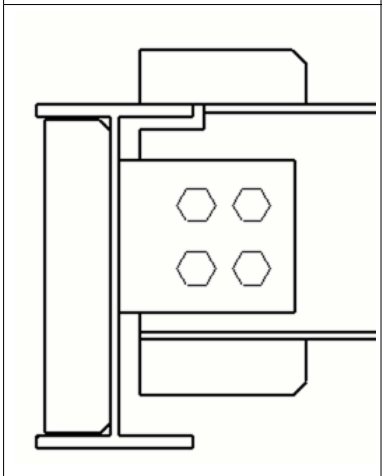
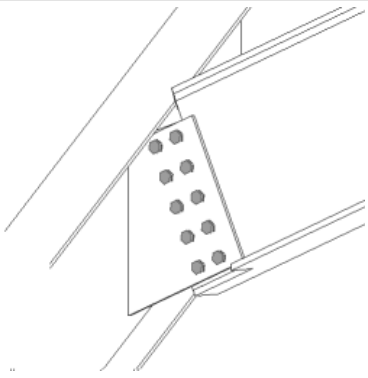
secondary beam can be leveled or sloped and/or skewed. A stiffener plate on the opposite side of the main beam web is optional.

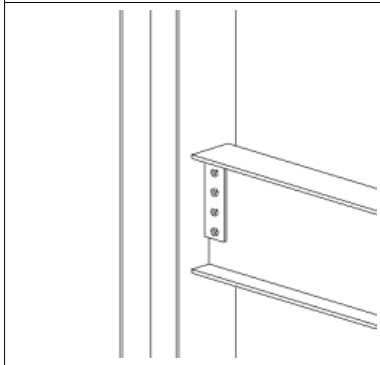
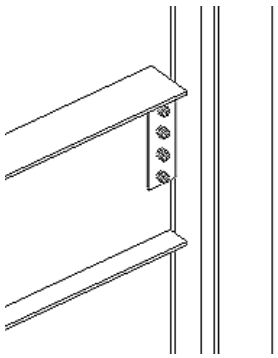
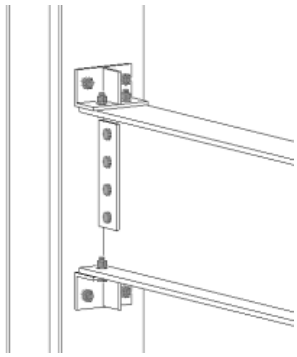
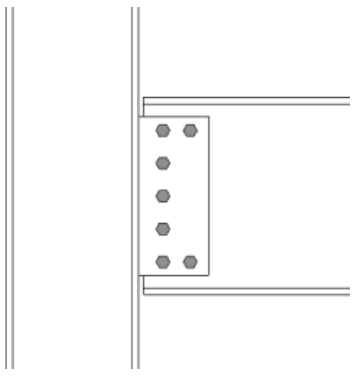
Objects created

- Shear tab (1 or 2)
- Stiffener (optional)
- Haunch plates (optional)
- Weld backing bars (optional)
- Seat angles
- Welds
- Bolts
- Cuts

Use for

Situation	Description
	Simple shear tab connected to a beam.
	Simple shear tab connected to a beam. Some bolts have been deleted.

Situation	Description
	Simple shear tab connected to a beam. The secondary part is skewed.
	Simple shear tab connected to a beam. The secondary part is sloped and skewed. The bolts and shear tab are parallel with the secondary part.
	Simple shear tab connected to a beam with haunches and a stiffener plate.
	Simple shear tab connected to a beam. The secondary part is sloped and skewed.

Situation	Description
	Simple shear tab connected to a column flange.
	Simple shear tab connected to a column flange/ edge.
	Simple shear tab connected to a column flange with seat angle options.
	Simple shear tab connected to a column flange. Some bolts have been deleted.

Selection order

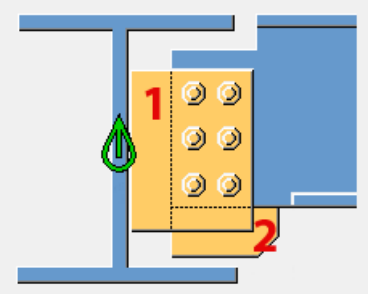
1. Select the main part (column or beam).

2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Part identification key

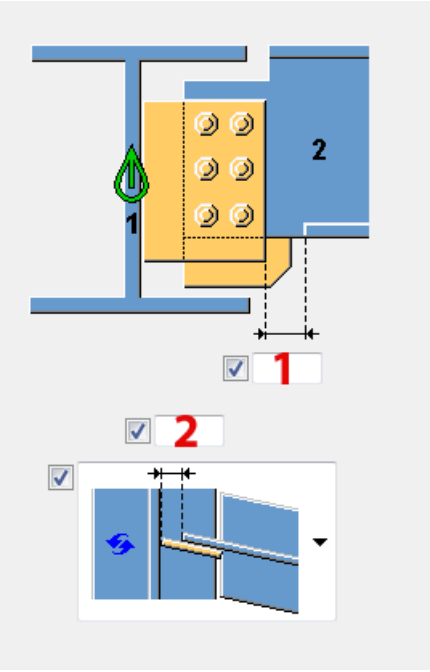


	Part
1	Shear tab
2	Haunch plate

Picture tab

Use the **Picture** tab to control the beam end, flange, and web cuts.

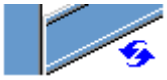
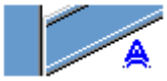
Dimensions



	Description	Default
1	Size of the strip made to the secondary part flange. The cut of the flange is defined from the shear tab edge.	The flange is automatically stripped when the shear tab crosses the flange. 20 mm
2	The beam gap dimension adjusts the gap between the main part web and the secondary beam web. The dimension is measured either square to the main part or in the same direction as the secondary part. This option applies in square and skewed framing conditions. The dimension is only used when the **Beam end cut** option is set to **default** or **automatic**.	20 mm Square to the main part

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.
	Square Cuts the end of the secondary beam square.
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.
	Square cut closer to the main part web Cuts the end of the secondary beam square and places the beam closer to the main part web.

Option	Description
	Clipped flange Cuts the corner of the flange at the end of the secondary beam.

Beam web cut

Define how the secondary beam web end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the web bevel when the end of the secondary beam is cut bevel.
	Square Cuts the end of the web square even if the end of the secondary beam is cut bevel.

Beam flange cut

Define how the secondary beam flange end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the flange bevel.
	Square Cuts a part of the flange square and leaves a part of it bevel.

Beam bottom flange cut

Option	Description
	Default Flange cut AutoDefaults can change this option.
	Notch The bottom of the secondary beam is notched if the shear tab crosses the flange. Enter the notch radius and height.
	Flange cut The secondary beam flange is cut on the same side as the shear tab if the shear tab crosses the flange.

Plates tab

Use the **Plates** tab to control the size, position, number, orientation and shape of the shear tab.

Parts

Option	Description
Tab plate	Shear tab plate thickness, width and height.
Plate washer	Plate washer thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

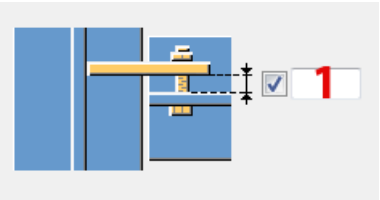
Option	Description	Default
Finish	Describes how the part surface has been treated.	

Shear tab position

Define the number and the side of shear tabs in single shear tab connections.

Option	Description
	Default Far side shear tab AutoDefaults can change this option.
	Automatic The component automatically selects either the near side or the far side shear tab. The tab is created to the side of the secondary part when the angle between the main part and the secondary part is less than 90 degrees.
	Far side shear tab
	Near side and far side shear tab
	Near side shear tab

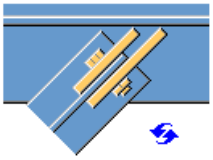
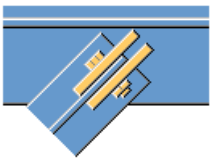
Gap between shear tab and secondary part



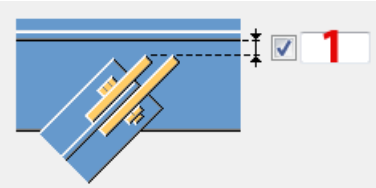
1	Gap between the secondary part web and the shear tab.	0

Shear tab position (sloped)

Define the position of shear tabs. The secondary part is sloped.

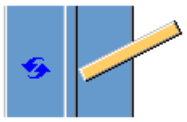
Option	Description
	Default Shear tab edges reach the same level in the main part. AutoDefaults can change this option.
	Shear tab edges reach the same level in the main part.
	Shear tabs have the same length.

Shear tab edge distance

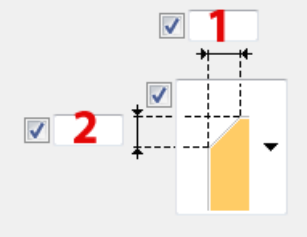


	Description
1	Distance between the main part web and the edge of the shear tabs.

Shear tab end cut

Option	Description
	Default The shear tab end is not cut. AutoDefaults can change this option.
	Square The shear tab end is not cut.
	Bevel The shear tab end is cut parallel to the main part web.

Shear tab chamfers



	Description
1	Horizontal dimension of the shear tab chamfer.
2	Vertical dimension of the shear tab chamfer.

Chamfer type

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

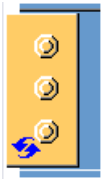
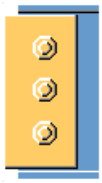
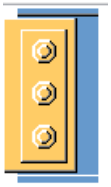
Shear tab orientation

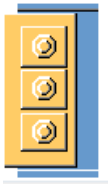
Option	Description
	Default Sloped AutoDefaults can change this option.

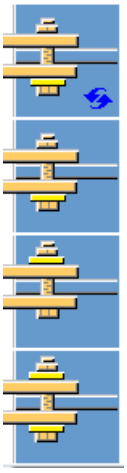
Option	Description
	Automatic The shear tab is sloped in the direction of the secondary beam. Both vertical edges of the shear tab are cut parallel to the end of the secondary beam.
	Sloped The shear tab is sloped in the direction of the secondary beam. Both vertical edges of the shear tab are cut parallel to the end of the secondary beam.
	Square
	Modified sloped Same as the **Sloped** option, but the vertical edge of the shear tab connected to the secondary beam is cut perpendicular to the secondary beam flange.

Plate washer

Define plate washers for bolts and select the plate washer side.

Option	Description
	Default No plate washer AutoDefaults can change this option.
	No plate washer
	One plate washer

Option	Description
	Individual square plate washers for each bolt
	Individual round plate washers for each bolt

Option	Description
	Select whether the plate washer is created for one shear tab or both shear tabs.

Stiffeners tab

Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position and type.

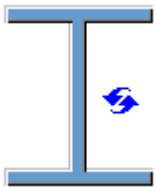
Opposite web stiffener plate dimensions

Option	Description
Opposite web stiffener	Opposite web stiffener plate thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

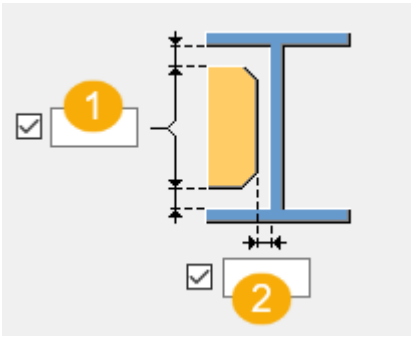
Option	Description	Default
	assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Stiffener creation

Option	Description
	Default No stiffeners are created. AutoDefaults can change this option.
	Full Creates a full stiffener of the same height as the web of the main part.
	Determined by shear tab Tekla Structures determines the size of the stiffener based on the shear tab size. Tekla Structures attempts to keep the bottom edges of the stiffener plate and shear tab level, if possible.
	Partial Leaves a gap between the stiffener plate and the bottom flange of the main part.

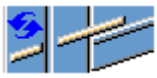
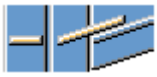
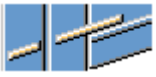
Option	Description
	No stiffeners are created.

Stiffener gap

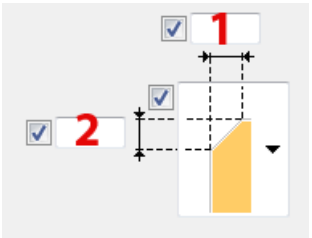


	Description
1	Size of the gap between the main part flanges and the stiffener.
2	Size of the gap between the main part web and the stiffener.

Stiffener orientation

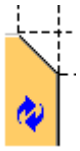
Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are perpendicular to the main part.
	Stiffeners are parallel to the secondary part.

Stiffener chamfers



	Description
1	Horizontal dimension of the chamfer.
2	Vertical dimension of the chamfer.

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Haunch tab

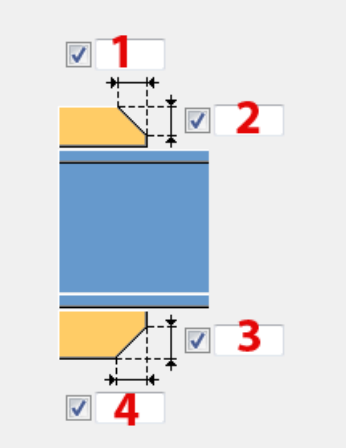
Use the **Haunch** tab to control the haunch plate creation and chamfers in the secondary beam flanges.

Haunch plates

Option	Description
Top plate	Top haunch plate thickness, width and height.
Bottom plate	Bottom haunch plate thickness, width and height.

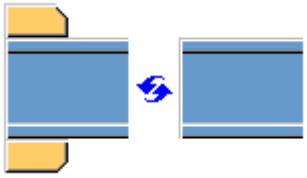
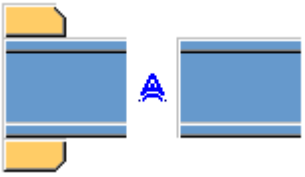
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Haunch plate chamfers



	Description
1	Width of the top haunch plate chamfer.
2	Height of the top haunch plate chamfer.
3	Height of the bottom haunch plate chamfer.
4	Width of the bottom haunch plate chamfer.

Haunch plate creation

Option	Description
	Default Top and bottom haunch plates are created, if needed. AutoDefaults can change this option.
	Automatic Top or bottom haunch plate or both are created, if needed.
	Top and bottom haunch plates are created. To create a single plate, enter `0` in the thickness (**t**) field for the plate you do not need (top or bottom plate).
	Haunch plates are not created.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

Automatic notching options affect both the top and the bottom flange.

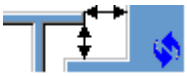
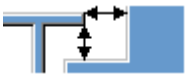
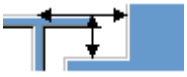
Notch shape

Automatic notching is switched on when you select a notch shape.

Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.

Option	Description
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.
	Creates notches on both sides of the secondary beam. The cuts are square to the secondary beam.

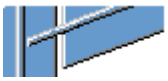
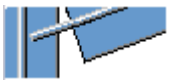
Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

Enter the horizontal and vertical values for the cuts.

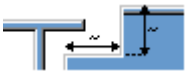
Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.

Option	Description
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.

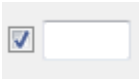
Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.



Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

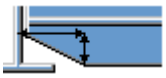
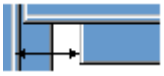
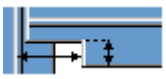
Side of flange notch

The side of flange notch defines on which side of the beam the notches are created.

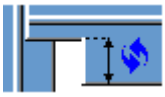
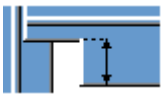
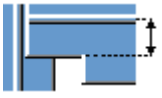
Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

Flange notch shape

The flange notch shape defines the notch shape in the beam flange.

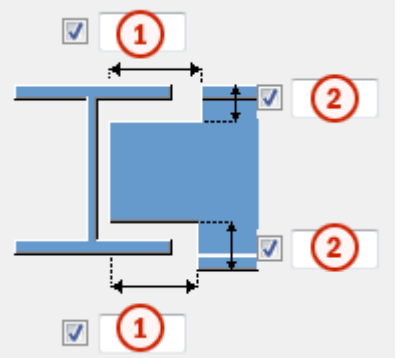
Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

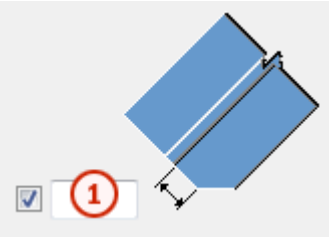
Enter the value for flange notch depth.

Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	`10 mm`
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest `5 mm`.

Dimension from web to flange cut



	Description
1	Define the distance between the web and the flange cut.

BCSA notch definition

Define whether the notch is created according to British Constructional Steelwork Association (BCSA) specifications.

Option	Description
Default	Notch dimensions.
Yes	Creates a 50 mm notch for simple beam-to-beam connections.

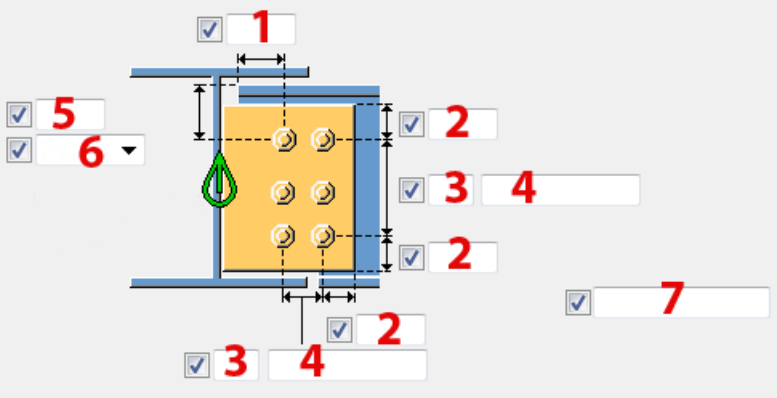
Option	Description
No	Use the options on this **Notch** tab to define the notch dimensions.

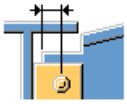
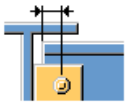
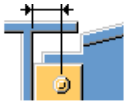
Bolts tab

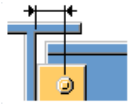
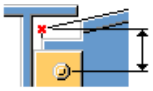
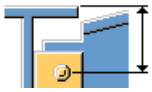
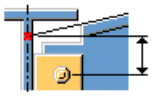
Use the **Bolts** tab to control the properties of the bolts that connect the shear tab to the secondary part.

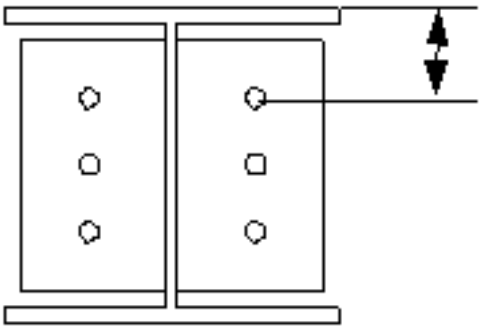
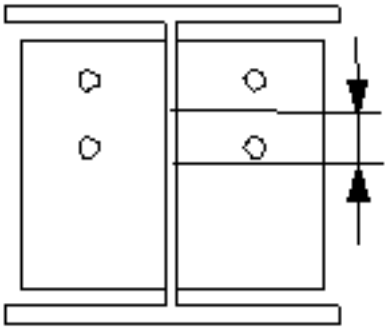
Bolt group dimensions

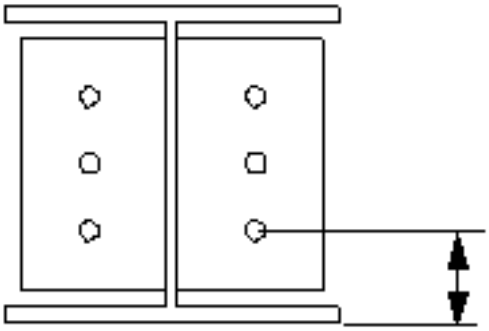
Bolt group dimensions affect the size and shape of the shear tab.



	Description
1	Dimension for horizontal bolt group position. When the secondary beam is sloped or skewed, define whether the horizontal dimension is measured from the bolt group to the edge of the secondary part, or from the bolt group to the main part web. • Sloped, to the secondary part  • Skewed, to the secondary part  • Sloped, to the main part 

	Description
	• Skewed, to the main part  The default is that the horizontal dimension is measured from the bolt group to the edge of the secondary part. Ensure that the **Beam end cut** on the **Picture** tab is set to square .
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position. When the secondary beam is sloped, define whether the vertical dimension is measured from the bolt group to the edge of the secondary part, or from the bolt group to the edge of the main part. • Vertical sloped dimension, to the secondary part • Vertical dimension, to the secondary part  • Vertical dimension, to the main part  • Vertical dimension, to the center line of the main part  The default is that the vertical dimension is measured from the bolt group the edge of the secondary part (sloped dimension).

	Description
	Ensure that the **Beam end cut** on the **Picture** tab is set to square .
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part. 

	Description
	• **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt group orientation

Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Staggered Bolts are staggered in the direction of the secondary part.
	Square Square bolt group is positioned horizontally.
	Sloped Square bolt group is sloped in the direction of the secondary part.

Bolting direction

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1
	Bolting direction 2

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	

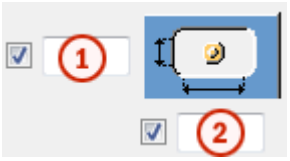
Option	Description	Default
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Slotted holes

You can define slotted, oversized, or tapped holes.

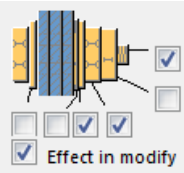


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Beam cut tab

Use the **Beam cut** tab to control weld backing bars, weld access holes, beam end preparations, and flange cuts.

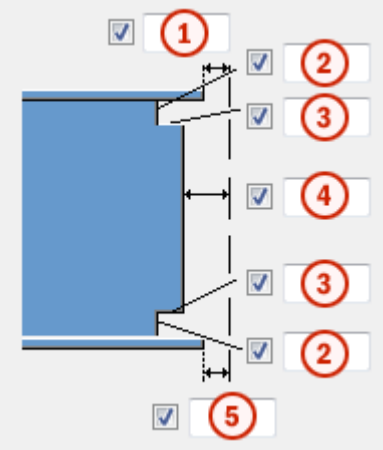
Weld backing bar

Option	Description
Weld backing bar	Weld backing bar thickness and width.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in

Option	Description	Default
		File menu --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

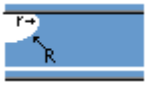
Weld access hole dimensions



	Description
1	Gap between the secondary part top flange and the main part.
2	Vertical dimensions for the top and the bottom weld access holes.
3	Horizontal dimensions for the top and the bottom weld access holes.
4	Gap between the secondary part web and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.
5	Gap between the secondary part bottom flange and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.

Weld access hole types

Option	Description	Default
	Default Round weld access hole AutoDefaults can change this option.	

Option	Description	Default
	Round weld access hole	
	Square weld access hole	
	Diagonal weld access hole	
	Round weld access hole with a radius that you can define in r ☑	
	Extended cone-shaped weld access hole with a radius and dimensions that you can define in R ☑ and Top Prep x ☑ Bottom Prep x ☑	
	Cone-shaped weld access hole with radiuses that you can define in R ☑ and r ☑ Capital **R** defines the large radius (height). Small **r** defines the small radius.	`R = 35` `r = 10`

Beam end preparation

Option	Description
	Default Top and bottom flange are prepared. AutoDefaults can change this option.

Option	Description
	Automatic Top and bottom flange are prepared.
	Beam end is not prepared.
	Top and bottom flange are prepared.
	Top flange is prepared.
	Bottom flange is prepared.

Flange cut

Option for top flange	Option for bottom flange	Description
		Default Flange is not cut. AutoDefaults can change this option.
		Flange is not cut.
		Flange is cut.

Weld backing bar creation

Option for bottom backing bar	Description
	Default Backing bars are created inside the flanges. AutoDefaults can change this option.
	No backing bars are created.
	Backing bars are created inside the flanges.

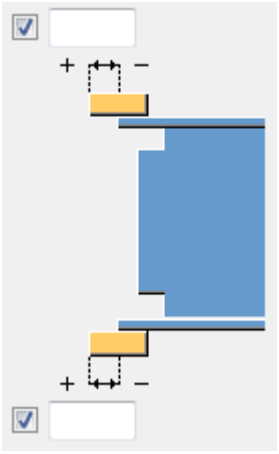
Option for bottom backing bar	Description
	Backing bars are created outside the flanges.

Weld backing bar length

Enter the length of the weld backing bar in the box below the options.

Option	Description
	Default Absolute length of the backing bar AutoDefaults can change this option.
	Absolute length of the backing bar
	Extension beyond the edge of the flange

Weld backing bar position

Option	Description
	Enter a positive or a negative value to move the front end of the backing bar relative to the end of the flange.

Assembly type

Define the location where the weld backing bar welds are made. When you select the **Workshop** option, Tekla Structures includes the backing bars in the assembly.

Angle box tab

Use the **Angle box** tab to add a seat angle.

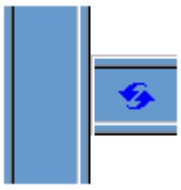
Seat angle

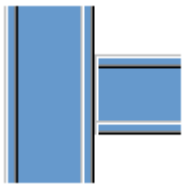
The purpose of seat angles is to carry loads from the secondary part. Seat angles can be positioned to top, bottom or both flanges of the secondary part. The seat angle can be stiffened, and bolted or welded to the main and secondary parts.

Option	Description
Stiffeners	Stiffener thickness, width and height.
Profile	Seat angle profile by selecting it from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

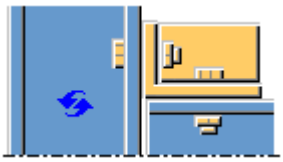
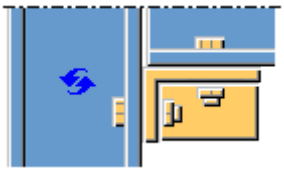
Seat angle position

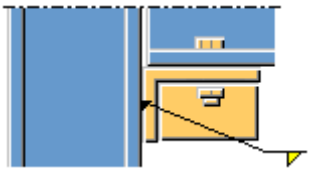
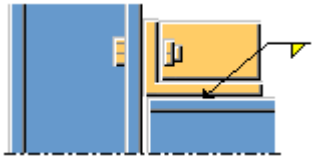
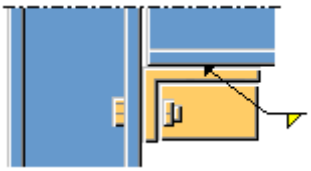
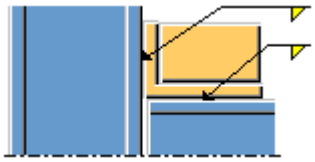
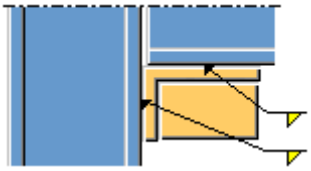
Option	Description
	Default No seat angle is created. AutoDefaults can change this option.

Option	Description
	No seat angle is created.
	Seat angle is created at the top of the flange.
	Seat angle is created at the bottom of the flange.
	Seat angles are created on both sides of the flange.

Seat angle attachment

Seat angle is positioned at the top or at the bottom of the secondary part.

Option for top seat angle	Option for bottom seat angle	Description
		Default Bolted Seat angle is bolted to the main part and to the secondary part. AutoDefaults can change this option.
		Bolted Seat angle is bolted to the main part and to the secondary part.

Option for top seat angle	Option for bottom seat angle	Description
		Welded-bolted Seat angle is welded to the main part and bolted to the secondary part.
		Bolted-welded Seat angle is bolted to the main part and welded to the secondary part.
		Welded Seat angle is welded to the main part and to the secondary part.

Seat angle stiffener type

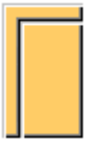
Option	Description
	Default Rectangular stiffener plate AutoDefaults can change this option.
	Rectangular stiffener plate
	Triangular stiffener plate
	The line connecting the ends of the seat angle legs defines the stiffener plate shape.

Seat angle rotation

Option	Description
	Default Seat angle is not rotated. AutoDefaults can change this option.

Option	Description
	Seat angle is not rotated.
	Seat angle is rotated horizontally 90 degrees. To stiffen the rotated seat angle, select the **Middle stiffeners** option in the **Middle stiffener position** list.

Seat angle orientation

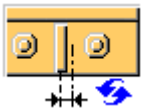
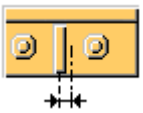
Option	Description
	Default The longer leg of the seat angle is connected to the secondary part. AutoDefaults can change this option.
	The longer leg of the seat angle is connected to the secondary part.
	The longer leg of the seat angle is connected to the main part.
	Automatic The longer leg of the seat angle is connected to the part where bolts reach furthest from the seat angle corner.

Seat angle side stiffener position

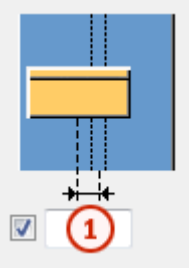
Option	Description
	Default No side stiffeners are created. AutoDefaults can change this option.
	No side stiffeners are created.
	Near side stiffeners are created.

Option	Description
	Far side stiffeners are created.
	Near side and far side stiffeners are created.

Seat angle middle stiffener position

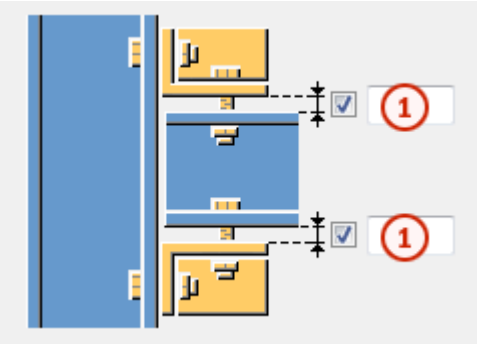
Option	Description
	Default According to bolts AutoDefaults can change this option.
	No middle stiffener plate is created.
	Middle stiffeners The stiffener plate is positioned in the middle of the seat angle. Enter the number of middle stiffeners in the **Number of middle stiffeners** box. Multiple stiffeners are centered and equally spaced.
	According to bolts The stiffener plate is positioned between the bolts in the middle of the bolt spacing. By default, stiffener is created between every two bolts. Enter the number of middle stiffeners in the box below the **According to bolts** option.

Seat angle offset



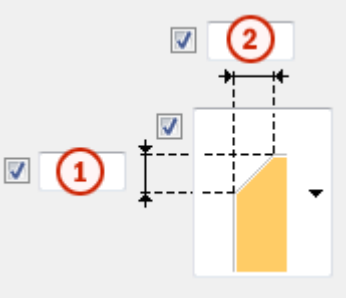
	Description
1	Seat angle horizontal offset from the center line of the main part.

Seat angle gap



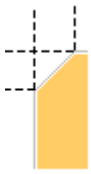
	Description
1	Top gap and bottom gap between the seat angle and the secondary part.

Seat angle chamfers



	Description
1	Vertical dimension of the chamfer.
2	Horizontal dimension of the chamfer.

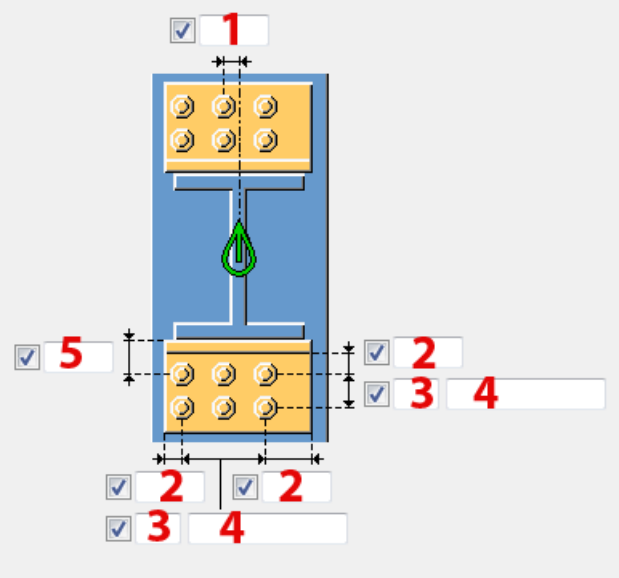
Chamfer type

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

BoxPBolts tab

Use the **BoxPBolts** tab to control properties of the bolts that connect the seat angle to the main part.

Seat angle bolt group dimensions



	Description
1	Dimension for horizontal bolt group position. The dimension is defined from the middle line of the secondary beam.
2	Edge distance is the distance from the center of a bolt to the edge of the part. Bolt edge distance.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position. The dimension is defined from the bottom of the secondary beam.

Bolt basic properties

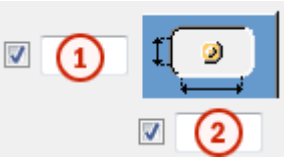
Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	

Option	Description	Default
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

- **Top** refers to the bolt group that connects the top seat angle to the main part.
- **Bottom** refers to the bolt group that connects the bottom seat angle to the main part.

Slotted holes

You can define slotted, oversized, or tapped holes.

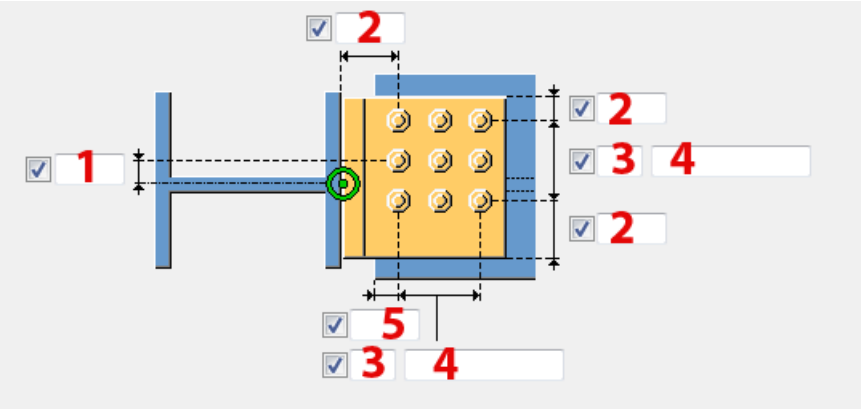


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

BoxSBolts tab

Use the **BoxSBolts** tab to control the properties of the bolts that connect the seat angle to the secondary part.

Seat angle bolt group dimensions



	Description
1	Dimension for vertical bolt group position. The dimension is defined from the middle line of the secondary beam.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for horizontal bolt group position. The dimension is defined from the bottom of the secondary beam.

Bolt basic properties

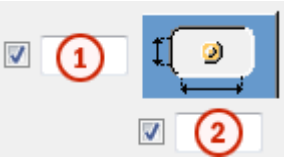
Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	

Option	Description	Default
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

- **Top** refers to the bolt group that connects the top seat angle to the secondary part.
- **Bottom** refers to the bolt group that connects the bottom seat angle to the secondary part.

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

General tab

Click the link below to find out more:

General tab

Design type tab

Click the link below to find out more:

Design type tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

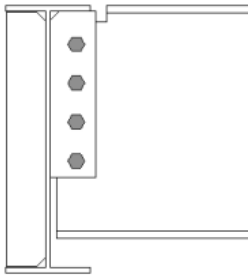
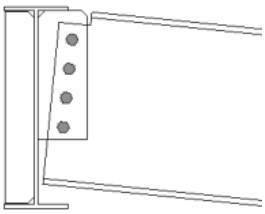
Welded to top flange (147)

Welded to top flange (147) connects two beams with a single shear tab or double shear tabs. The shear tabs are welded to the main beam web and the top flange, and bolted to secondary beam web. The secondary beam can be leveled or sloped.

Objects created

- Shear tab (1 or 2)
- Stiffeners (optional)
- Haunch plates (optional)
- Weld backing bars (optional)
- Seat angles (optional)
- Bolts
- Welds
- Cuts

Use for

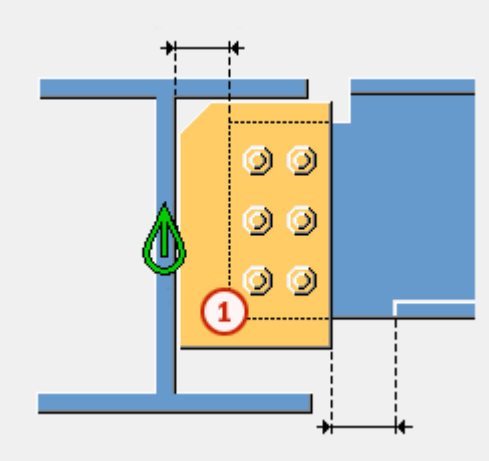
Situation	Description
	Partial depth shear tab connected to the top flange of a beam. Stiffener is created.
	Partial depth shear tab connected to the top flange of a beam. The secondary part is sloped. Stiffener is created.

Selection order

1. Select the main part (beam).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

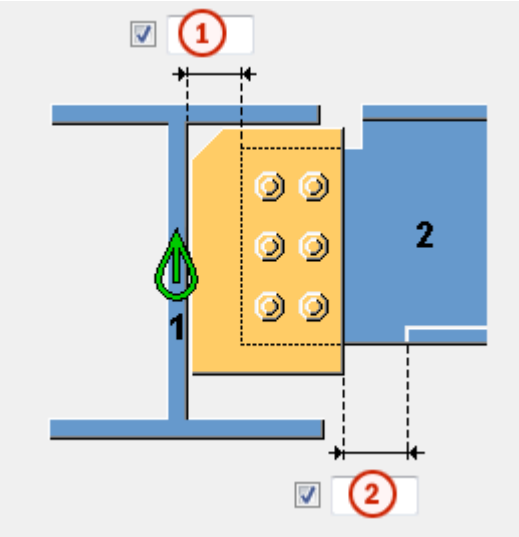


	Part
1	Shear tab

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

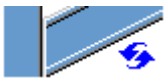
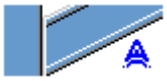
Use the **Picture** tab to control the position of the shear tab, and the beam flange and the web cuts.



	Description	Default
1	Cut of the secondary part. Cutting the secondary part creates a gap between the main part and the secondary part.	`20`
2	Size of the strip made to the secondary part flange. The cut of the flange is defined from the shear tab edge.	The flange is automatically stripped when the shear tab crosses the flange. `20`

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.
	Square Cuts the end of the secondary beam square.
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.
	Square cut closer to the main part web Cuts the end of the secondary beam square and places the beam closer to the main part web.
	Clipped flange Cuts the corner of the flange at the end of the secondary beam.

Beam web cut

Define how the secondary beam web end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the web bevel when the end of the secondary beam is cut bevel.
	Square Cuts the end of the web square even if the end of the secondary beam is cut bevel.

Beam flange cut

Define how the secondary beam flange end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the flange bevel.
	Square Cuts a part of the flange square and leaves a part of it bevel.

Beam bottom flange cut

Option	Description
	Default Flange cut AutoDefaults can change this option.
	Notch The bottom of the secondary beam is notched if the shear tab crosses the flange. Enter the notch radius and height.
	Flange cut The secondary beam flange is cut on the same side as the shear tab if the shear tab crosses the flange.

Plates tab

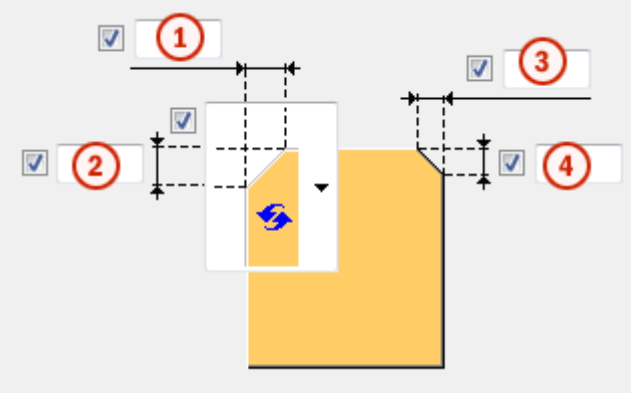
Use the **Plates** tab to control the size, position, number, orientation and shape of the shear tab.

Shear tab plate

Option	Description
Tab plate	Shear tab plate thickness, width and height.

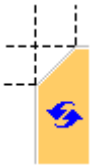
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

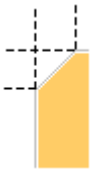
Shear tab chamfers



	Description
1	Horizontal dimension of the shear tab chamfer.
2	Vertical dimension of the shear tab chamfer.
3	Horizontal dimension of the shear tab exterior chamfer.
4	Vertical dimension of the shear tab exterior chamfer.

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.

Option	Description
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Shear tab position

Define the number and the side of shear tabs in single shear tab connections.

Option	Description
	Default Far side shear tab AutoDefaults can change this option.
	Automatic The component automatically selects either the near side or the far side shear tab. The tab is created to the side of the secondary part when the angle between the main part and the secondary part is less than 90 degrees.
	Far side shear tab

Option	Description
	Near side and far side shear tab
	Near side shear tab

Stiffeners tab

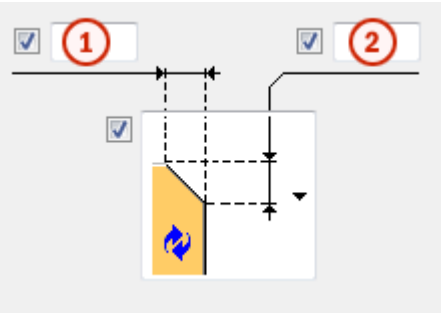
Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position and type.

Opposite web stiffener plate dimensions

Option	Description
Opposite web stiffener	Opposite web stiffener plate thickness, width and height.

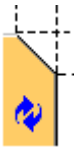
Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Chamfer dimensions

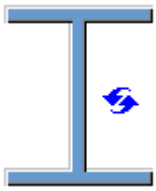


	Description
1	Horizontal dimension of the chamfer.
2	Vertical dimension of the chamfer.

Chamfer type

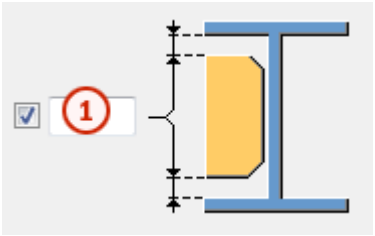
Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Stiffener creation

Option	Description
	Default No stiffeners are created. AutoDefaults can change this option.
	Full Creates a full stiffener of the same height as the web of the main part.

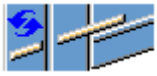
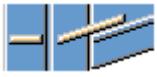
Option	Description
	Determined by shear tab Tekla Structures determines the size of the stiffener based on the shear tab size. Tekla Structures attempts to keep the bottom edges of the stiffener plate and shear tab level, if possible.
	Partial Leaves a gap between the stiffener plate and the bottom flange of the main part.
	No stiffeners are created.

Stiffener gap



	Description
1	Size of the gap between the main part flanges and the stiffener.

Stiffener orientation

Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are perpendicular to the main part.
	Stiffeners are parallel to the secondary part.

Haunch tab

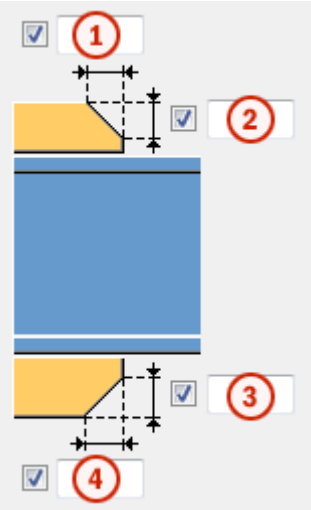
Use the **Haunch** tab to control the haunch plate creation and chamfers in the secondary beam flanges.

Haunch plates

Option	Description
Top plate	Top haunch plate thickness, width and height.
Bottom plate	Bottom haunch plate thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

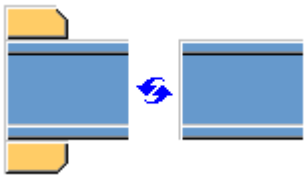
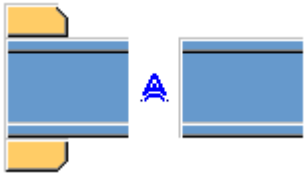
Haunch plate chamfers



	Description
1	Width of the top haunch plate chamfer.
2	Height of the top haunch plate chamfer.
3	Height of the bottom haunch plate chamfer.

	Description
4	Width of the bottom haunch plate chamfer.

Haunch plate creation

Option	Description
	Default Top and bottom haunch plates are created, if needed. AutoDefaults can change this option.
	Automatic Top or bottom haunch plate or both are created, if needed.
	Top and bottom haunch plates are created. To create a single plate, enter `0` in the thickness (**t**) field for the plate you do not need (top or bottom plate).
	Haunch plates are not created.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

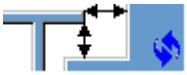
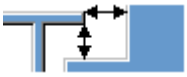
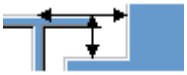
Automatic notching options affect both the top and the bottom flange.

Notch shape

Automatic notching is switched on when you select a notch shape.

Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

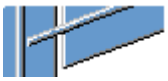
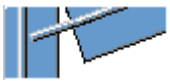
Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

Enter the horizontal and vertical values for the cuts.

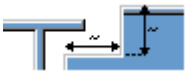
Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.

Option	Description
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.

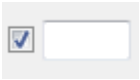
Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.



Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

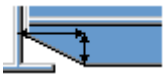
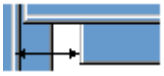
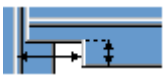
Side of flange notch

The side of flange notch defines on which side of the beam the notches are created.

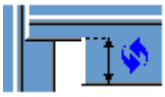
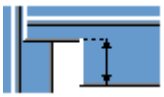
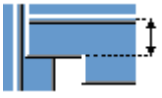
Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

Flange notch shape

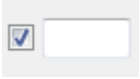
The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

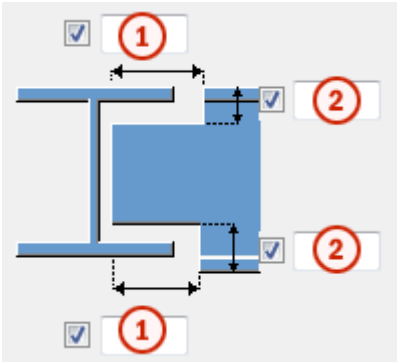
Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.

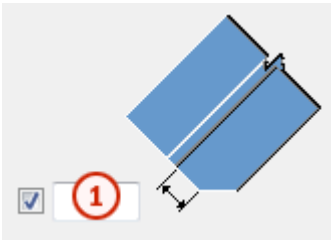


Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	`10 mm`
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest `5 mm`.

Dimension from web to flange cut

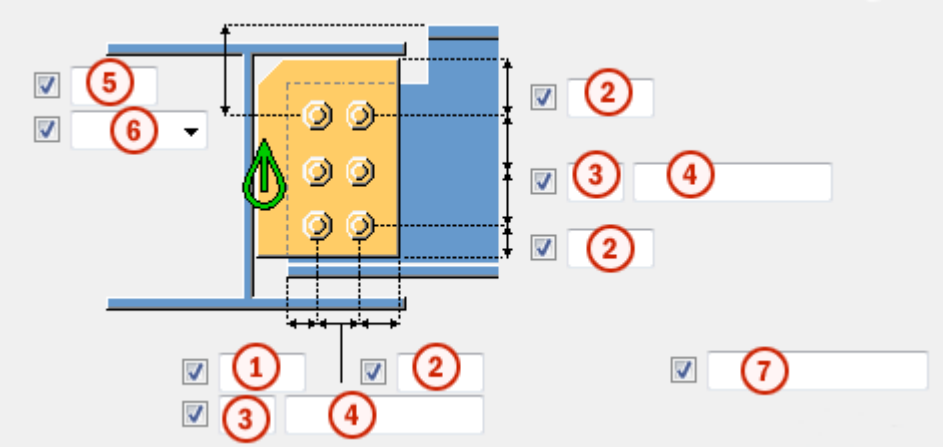


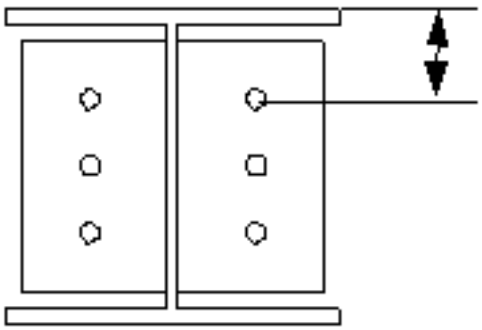
	Description
1	Define the distance between the web and the flange cut.

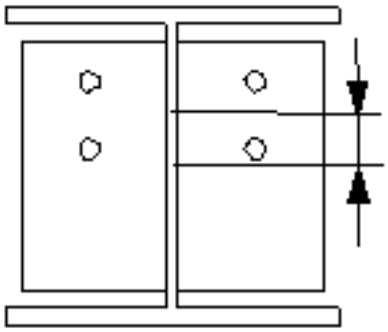
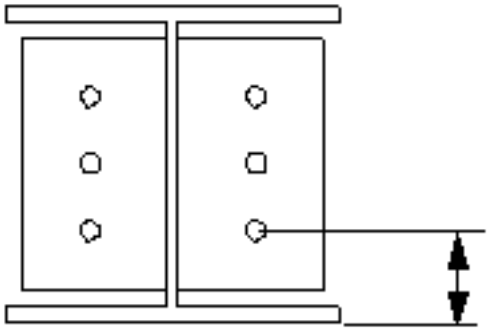
Bolts tab

Use the **Bolts** tab to control the properties of the bolts that connect the shear tab to the main part.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Dimension for vertical bolt group position.
5	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
6	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

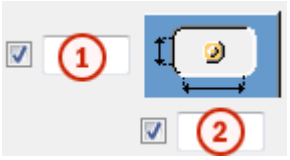
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within	`Yes`

Option	Description	Default
	the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

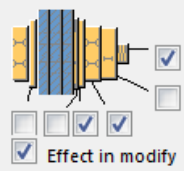


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt group orientation

Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square

Option	Description
	Staggered Bolts are staggered in the direction of the secondary part.
	Square Square bolt group is positioned horizontally.
	Sloped Square bolt group is sloped in the direction of the secondary part.

Beam cut tab

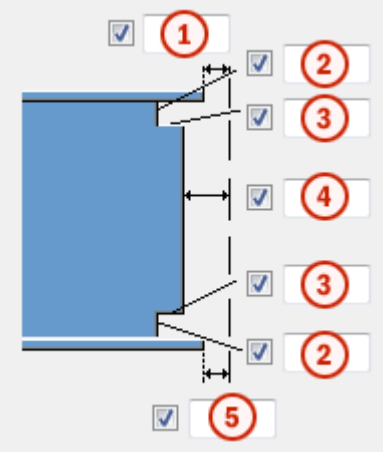
Use the **Beam cut** tab to control weld backing bars, weld access holes, beam end preparations, and flange cuts.

Weld backing bar

Option	Description
Weld backing bar	Weld backing bar thickness and width.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

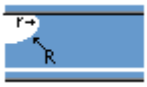
Weld access hole dimensions



	Description
1	Gap between the secondary part top flange and the main part.
2	Vertical dimensions for the top and the bottom weld access holes.
3	Horizontal dimensions for the top and the bottom weld access holes.
4	Gap between the secondary part web and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.
5	Gap between the secondary part bottom flange and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.

Weld access holes

Option	Description	Default
	Default Round weld access hole AutoDefaults can change this option.	
	Round weld access hole	
	Square weld access hole	
	Diagonal weld access hole	

Option	Description	Default
	Round weld access hole with a radius that you can define in r ☑	
	Extended cone-shaped weld access hole with a radius and dimensions that you can define in R ☑ and Top Prep x ☑ Bottom Prep x ☑	
	Cone-shaped weld access hole with radiuses that you can define in R ☑ and r ☑ Capital **R** defines the large radius (height). Small **r** defines the small radius.	R = 35 r = 10

Beam end preparation

Option	Description
	Default Top and bottom flange are prepared. AutoDefaults can change this option.
	Automatic Top and bottom flange are prepared.
	Beam end is not prepared.
	Top and bottom flange are prepared.

Option	Description
	Top flange is prepared.
	Bottom flange is prepared.

Flange cut

Option for top flange	Option for bottom flange	Description
		Default Flange is not cut. AutoDefaults can change this option.
		Flange is not cut.
		Flange is cut.

Weld backing bars

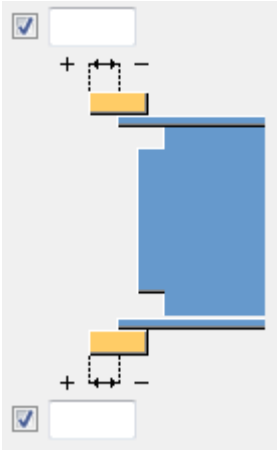
Option for bottom backing bar	Description
	Default Backing bars are created inside the flanges. AutoDefaults can change this option.
	No backing bars are created.
	Backing bars are created inside the flanges.
	Backing bars are created outside the flanges.

Weld backing bar length

Enter the length of the weld backing bar in the box below the options.

Option	Description
	Default Absolute length of the backing bar AutoDefaults can change this option.
	Absolute length of the backing bar
	Extension beyond the edge of the flange

Weld backing bar position

Option	Description
	Enter a positive or a negative value to move the front end of the backing bar relative to the end of the flange.

Assembly type

Define the location where the weld backing bar welds are made. When you select the **Workshop** option, Tekla Structures includes the backing bars in the assembly.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

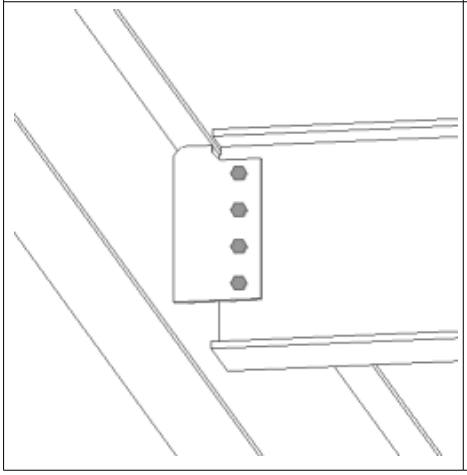
Welded to top flange S (149)

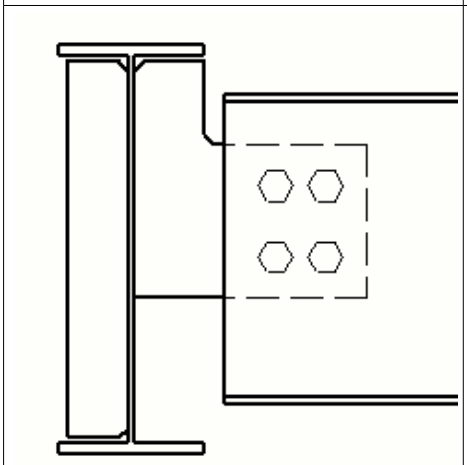
Welded to top flange S (149) connects two beams with a single shear tab or double shear tab. The shear tabs are welded to the main beam web and the top flange, and bolted to secondary beam web. The secondary beam can be leveled or sloped.

Objects created

- Shear tab (1 or 2)
- Stiffener (optional)
- Haunch plates (optional)
- Weld backing bars (optional)
- Bolts
- Welds
- Cuts

Use for

Situation	Description
	Partial depth shear tab connected to the top flange of a beam.

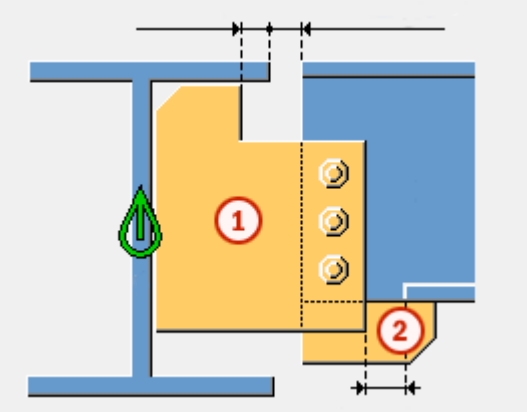
Situation	Description
	Partial depth shear tab connected to the top flange of a beam. Stiffener is created.

Selection order

1. Select the main part (beam).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key



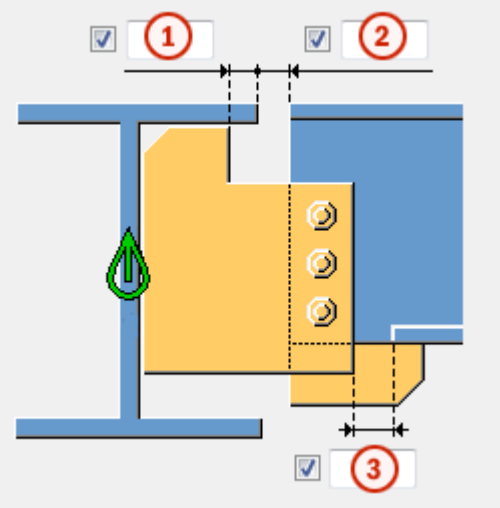
	Part
1	Shear tab
2	Haunch plate

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to control the position of the shear tab, and the beam flange and the web cuts.

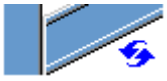
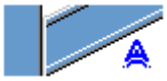
Dimensions



	Description	Default
1	Shear tab edge distance from the main part flange edge.	0
2	Cut of the secondary part. Cutting the secondary part creates a gap between the main part and the secondary part.	20
3	Size of the strip made to the secondary part flange. The cut of the flange is defined from the shear tab edge.	The flange is automatically stripped when the shear tab crosses the flange. 20

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut

Option	Description
	square. Otherwise, the beam end is cut bevel.
	Square Cuts the end of the secondary beam square.
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.
	Square cut closer to the main part web Cuts the end of the secondary beam square and places the beam closer to the main part web.
	Clipped flange Cuts the corner of the flange at the end of the secondary beam.

Beam web cut

Define how the secondary beam web end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the web bevel when the end of the secondary beam is cut bevel.
	Square Cuts the end of the web square even if the end of the secondary beam is cut bevel.

Beam flange cut

Define how the secondary beam flange end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the flange bevel.
	Square Cuts a part of the flange square and leaves a part of it bevel.

Beam bottom flange cut

Option	Description
	Default Flange cut AutoDefaults can change this option.
	Notch The bottom of the secondary beam is notched if the shear tab crosses the flange. Enter the notch radius and height.
	Flange cut The secondary beam flange is cut on the same side as the shear tab if the shear tab crosses the flange.

Plates tab

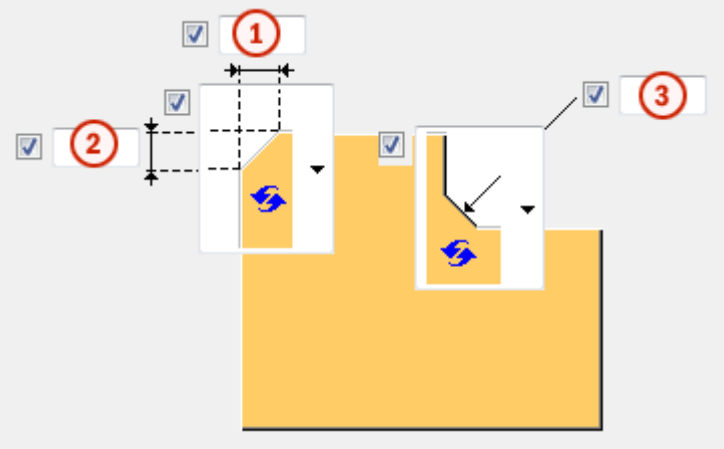
Use the **Plates** tab to control the size, position, number, orientation and shape of the shear tab.

Shear tab plate

Option	Description
Tab plate	Shear tab plate thickness, width and height.

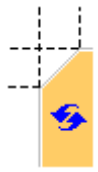
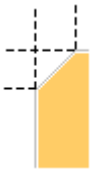
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Shear tab chamfers



	Description
1	Horizontal dimension of the shear tab chamfer.
2	Vertical dimension of the shear tab chamfer.
3	Vertical and the horizontal dimension of the shear tab chamfer.

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Chamfer type dimensions

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer

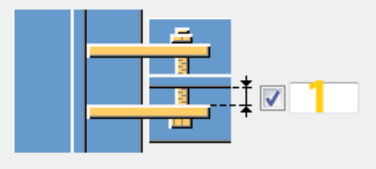
Option	Description
	Line chamfer
	Concave arc chamfer

Shear tab position

Define the number and the side of shear tabs in single shear tab connections.

Option	Description
	Default Far side shear tab AutoDefaults can change this option.
	Automatic The component automatically selects either the near side or the far side shear tab. The tab is created to the side of the secondary part when the angle between the main part and the secondary part is less than 90 degrees.
	Far side shear tab
	Near side and far side shear tab
	Near side shear tab

Gap between shear tabs



	Description	Default
1	Gap between the secondary part web and shear tab. This only affects connections with two shear tabs.	0

Stiffeners tab

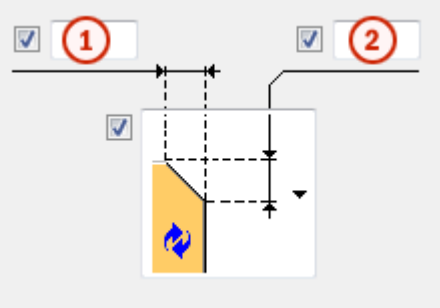
Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position and type.

Opposite web stiffener plate dimensions

Option	Description
Opposite web stiffener	Opposite web stiffener plate thickness, width and height.

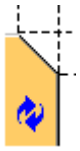
Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Chamfer dimensions

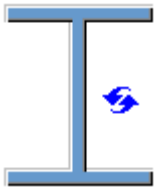


	Description
1	Horizontal dimension of the chamfer.
2	Vertical dimension of the chamfer.

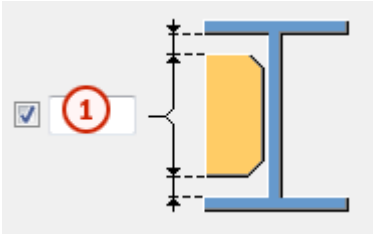
Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Stiffener creation

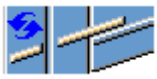
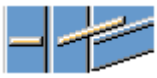
Option	Description
	Default No stiffeners are created. AutoDefaults can change this option.
	Full Creates a full stiffener of the same height as the web of the main part.
	Determined by shear tab Tekla Structures determines the size of the stiffener based on the shear tab size. Tekla Structures attempts to keep the bottom edges of the stiffener plate and shear tab level, if possible.
	Partial Leaves a gap between the stiffener plate and the bottom flange of the main part.
	No stiffeners are created.

Stiffener gap



	Description
1	Size of the gap between the main part flanges and the stiffener.

Stiffener orientation

Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are perpendicular to the main part.
	Stiffeners are parallel to the secondary part.

Haunch tab

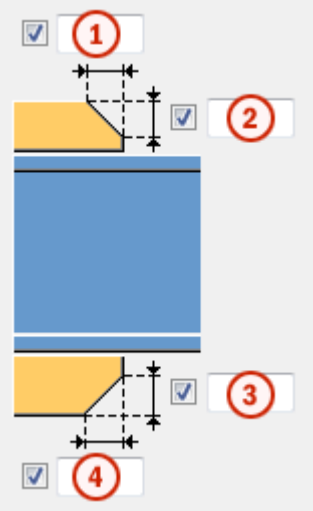
Use the **Haunch** tab to control the haunch plate creation and chamfers in the secondary beam flanges.

Haunch plates

Option	Description
Top plate	Top haunch plate thickness, width and height.
Bottom plate	Bottom haunch plate thickness, width and height.

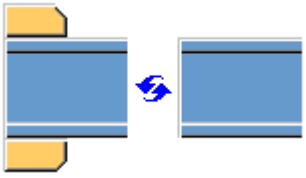
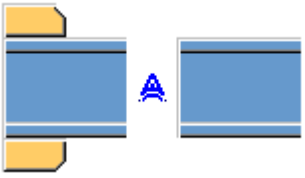
Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Haunch plate chamfers



	Description
1	Width of the top haunch plate chamfer.
2	Height of the top haunch plate chamfer.
3	Height of the bottom haunch plate chamfer.
4	Width of the bottom haunch plate chamfer.

Haunch plate creation

Option	Description
	Default Top and bottom haunch plates are created, if needed. AutoDefaults can change this option.
	Automatic Top or bottom haunch plate or both are created, if needed.
	Top and bottom haunch plates are created. To create a single plate, enter `0` in the thickness (**t**) field for the plate you do not need (top or bottom plate).

Option	Description
	Haunch plates are not created.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

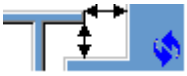
Automatic notching options affect both the top and the bottom flange.

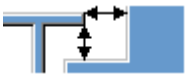
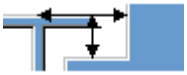
Notch shape

Automatic notching is switched on when you select a notch shape.

Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

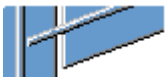
Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.

Option	Description
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

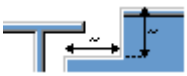
Enter the horizontal and vertical values for the cuts.

Flange cut shape

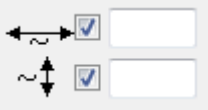
Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.



Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.

Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

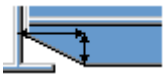
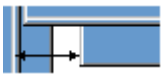
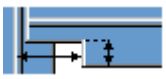
Side of flange notch

The side of flange notch defines on which side of the beam the notches are created.

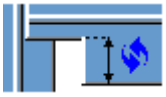
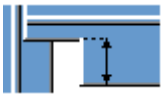
Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

Flange notch shape

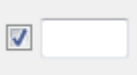
The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

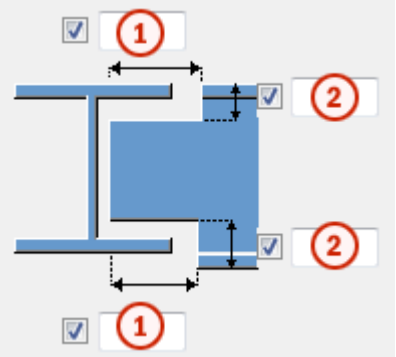
Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.

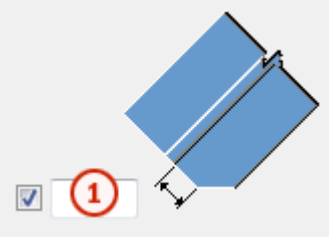


Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	`10 mm`
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest `5 mm`.

Dimension from web to flange cut

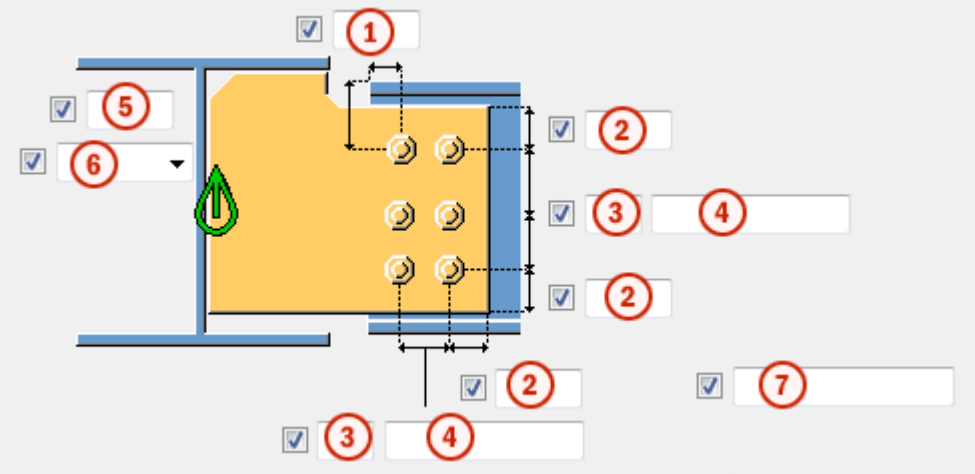


	Description
1	Define the distance between the web and the flange cut.

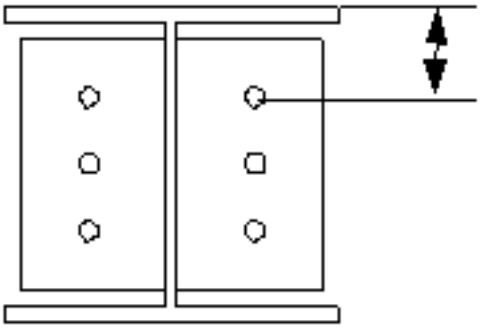
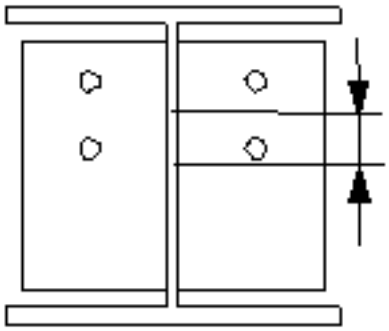
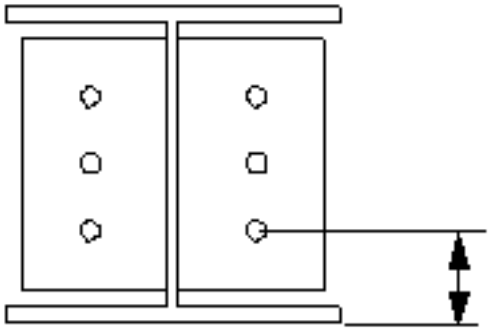
Bolts tab

Use the **Bolts** tab to control the properties of the bolts that connect the shear tab to the main part.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position.

	Description
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

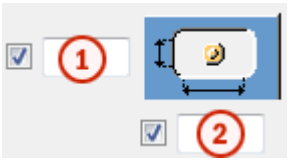
	Description
7	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



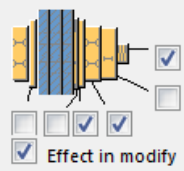
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered

Option	Description
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt group orientation

Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Staggered Bolts are staggered in the direction of the secondary part.
	Square Square bolt group is positioned horizontally.
	Sloped Square bolt group is sloped in the direction of the secondary part.

Bolting direction

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1
	Bolting direction 2

Beam cut tab

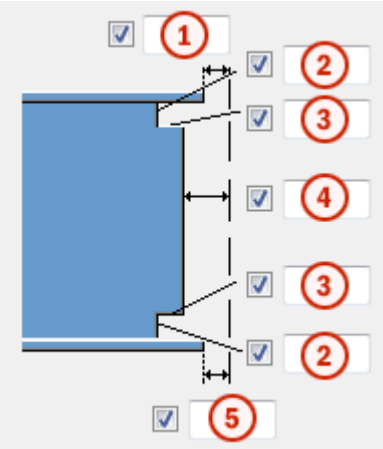
Use the **Beam cut** tab to control weld backing bars, weld access holes, beam end preparations, and flange cuts.

Weld backing bar

Option	Description
Weld backing bar	Weld backing bar thickness and width.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

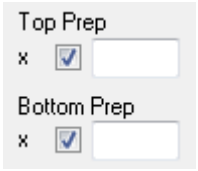
Weld access hole dimensions

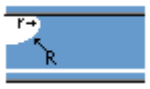
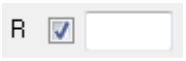


	Description
1	Gap between the secondary part top flange and the main part.

	Description
2	Vertical dimensions for the top and the bottom weld access holes.
3	Horizontal dimensions for the top and the bottom weld access holes.
4	Gap between the secondary part web and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.
5	Gap between the secondary part bottom flange and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.

Weld access holes

Option	Description	Default
	Default Round weld access hole AutoDefaults can change this option.	
	Round weld access hole	
	Square weld access hole	
	Diagonal weld access hole	
	Round weld access hole with a radius that you can define in 	
	Extended cone-shaped weld access hole with a radius and dimensions that you can define in and 	

Option	Description	Default
	Cone-shaped weld access hole with radiuses that you can define in  and Capital **R** defines the large radius (height). Small **r** defines the small radius.	R = 35 r = 10

Beam end preparation

Option	Description
	Default Top and bottom flange are prepared. AutoDefaults can change this option.
	Automatic Top and bottom flange are prepared.
	Beam end is not prepared.
	Top and bottom flange are prepared.
	Top flange is prepared.
	Bottom flange is prepared.

Flange cut

Option for top flange	Option for bottom flange	Description
		Default Flange is not cut. AutoDefaults can change this option.

Option for top flange	Option for bottom flange	Description
		Flange is not cut.
		Flange is cut.

Weld backing bars

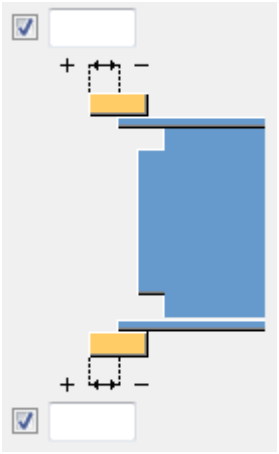
Option for bottom backing bar	Description
	Default Backing bars are created inside the flanges. AutoDefaults can change this option.
	No backing bars are created.
	Backing bars are created inside the flanges.
	Backing bars are created outside the flanges.

Weld backing bar length

Enter the length of the weld backing bar in the box below the options.

Option	Description
	Default Absolute length of the backing bar AutoDefaults can change this option.
	Absolute length of the backing bar
	Extension beyond the edge of the flange

Weld backing bar position

Option	Description
	Enter a positive or a negative value to move the front end of the backing bar relative to the end of the flange.

Assembly type

Define the location where the weld backing bar welds are made. When you select the **Workshop** option, Tekla Structures includes the backing bars in the assembly.

General tab

Click the links below to find out more:

General tab

Design tab

Click the links below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Moment connection (181)

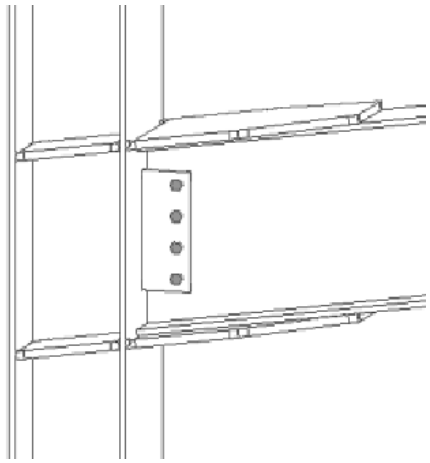
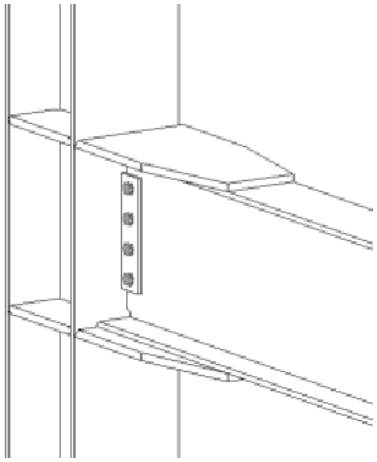
Moment connection (181) connects a beam to a column or to a beam with a single shear tab or double shear tabs. The shear tab is welded to the main part

and bolted to the secondary part web. The secondary beam can be leveled or sloped. Top and bottom flange plates may be welded to the secondary beam and site-welded to the main column.

Objects created

- Shear tab (1 or 2)
- Top flange plate
- Bottom flange plate
- Stiffeners (4) (optional)
- Web doubler plate (optional)
- Bolts
- Welds
- Cuts

Use for

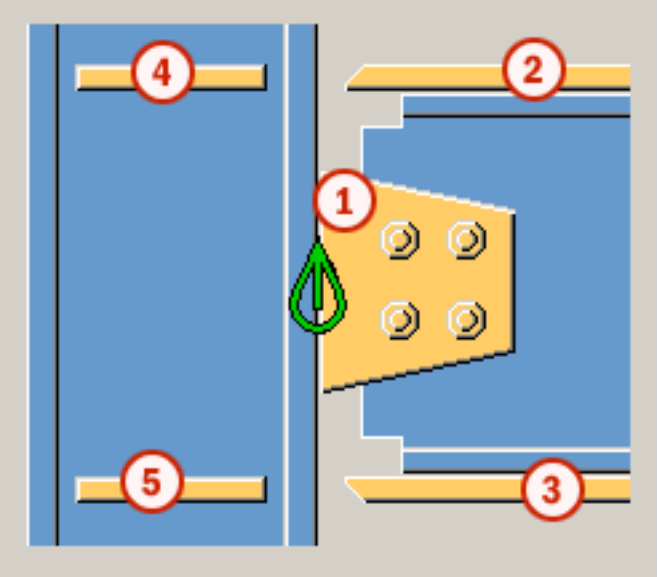
Situation	Description
	Moment connection welded to a column flange. The secondary part is sloped.
	Moment connection welded to a column flange. Beam weld preparation and weld access hole options.

Selection order

1. Select the main part (column or beam).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key



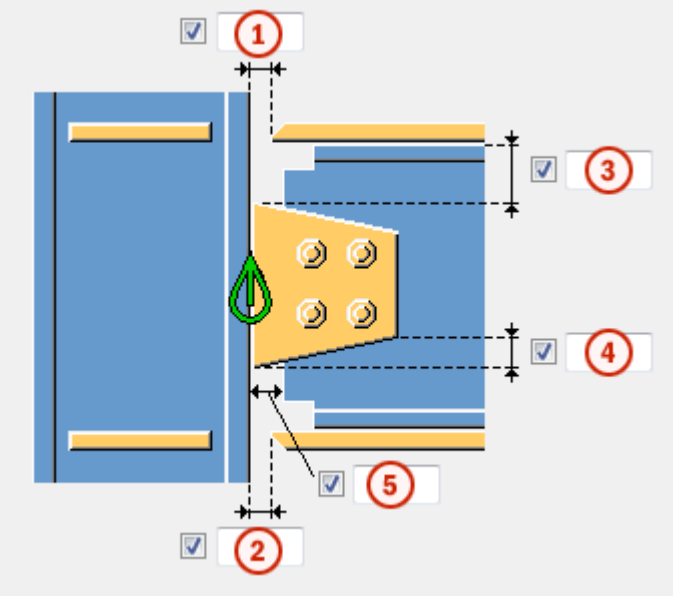
	Part
1	Shear tab
2	Top flange plate
3	Bottom flange plate
4	Top stiffener plate
5	Bottom stiffener plate

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to control the position of the shear tab, and the beam flange and the web cuts.

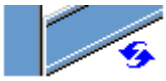
Dimensions



	Description
1	Site welding distance from the edge of the top flange plate to the main part flange.
2	Site welding distance from the edge of the bottom flange plate to the main part web.
3	Distance from the edge of the shear tab to the top edge of the secondary part.
4	Chamfer dimension of the shear tab.
5	Distance from the main part flange to the edge of the secondary part.

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.

Option	Description
	Square Cuts the end of the secondary beam square.
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.

Beam flange cut

Define how the secondary beam flange end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the flange bevel.
	Square Cuts a part of the flange square and leaves a part of it bevel.

Plates tab

Use the **Plates** tab to control the size, position, number, and shape of the shear tab.

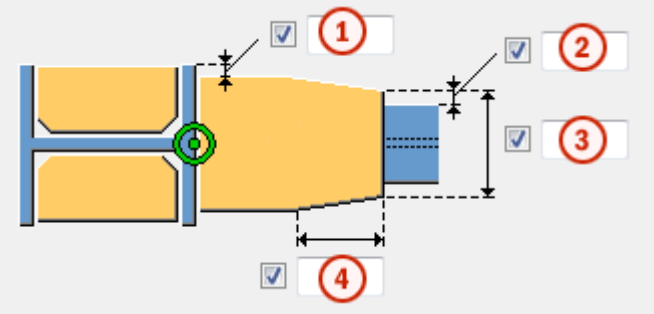
Plates

Option	Description
Tab plate	Shear tab plate thickness, width and height.

Option	Description
Top plate	Top plate thickness, width and height.
Bottom plate	Bottom plate thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Shear tab dimensions



	Description
1	Distance from the edge of the main part flange to the edge of the top and bottom plate.
2	Distance from the flange of the secondary part to the edge of the top and bottom plate.
3	Width of the chamfered end of the top and bottom plate.
4	Chamfering dimension of the top and bottom plate.

Shear tab position

Define the number and the side of shear tabs in single shear tab connections.

Option	Description
	Default Far side shear tab AutoDefaults can change this option.
	Automatic The component automatically selects either the near side or the far side shear tab. The tab is created to the side of the secondary part when the angle between the main part and the secondary part is less than 90 degrees.
	Far side shear tab
	Near side and far side shear tab
	Near side shear tab

Stiffeners tab

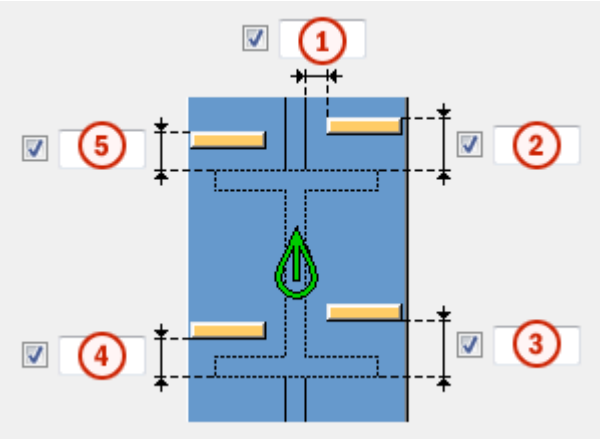
Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position and type.

Stiffener plate dimensions

Option	Description
Top NS	Top near side stiffener thickness, width and height.
Top FS	Top far side stiffener thickness, width and height.
Bottom NS	Bottom near side stiffener thickness, width and height.
Bottom FS	Bottom far side stiffener thickness, width and height.

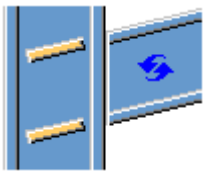
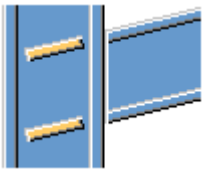
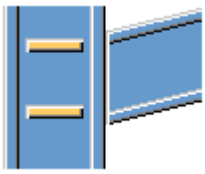
Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Stiffener positions

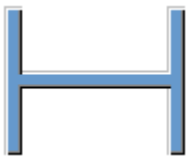


	Description
1	Size of the gap between the stiffener and the beam web edge.
2	Size of the gap between the top near side stiffener and the beam flange edge.
3	Size of the gap between the bottom near side stiffener and the beam flange edge.
4	Size of the gap between the bottom far side stiffener and the beam flange edge.
5	Size of the gap between the top far side stiffener and the beam flange edge.

Stiffener orientation

Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are parallel to the secondary part.
	Stiffeners are perpendicular to the main part.

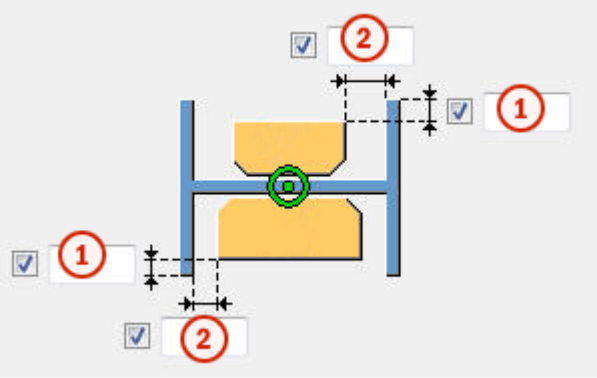
Stiffener creation

Option	Description
	Default Stiffeners are created. AutoDefaults can change this option.
	Automatic Stiffeners are created when necessary.
	No stiffeners are created.
	Stiffeners are created.

Stiffener shape

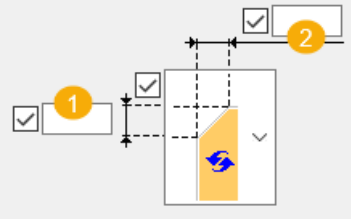
Option	Description
	Default Line chamfered stiffener plates AutoDefaults can change this option.
	Automatic Line chamfered stiffener plates
	Square stiffener plates Stiffener plates with a gap for the main part web rounding
	Line chamfered stiffener plates

Stiffener gap



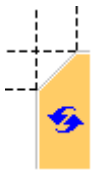
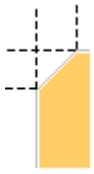
	Description
1	Distance from the edge of the flange to the edge of the stiffener.
2	Size of the gap between the flanges and the stiffener.

Chamfer dimensions



	Description	Default
1	Vertical dimension of the chamfer.	10 mm
2	Horizontal dimension of the chamfer.	10 mm

Chamfer type

Option	Description
	Default. Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections:

automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

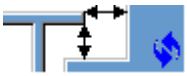
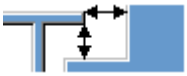
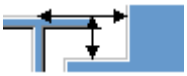
Automatic notching options affect both the top and the bottom flange.

Notch shape

Automatic notching is switched on when you select a notch shape.

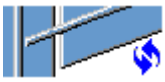
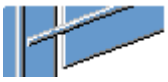
Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

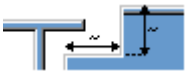
Enter the horizontal and vertical values for the cuts.

Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.

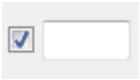
Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.



Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

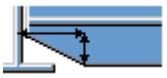
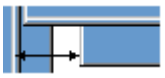
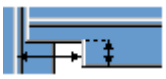
Side of flange notch

The side of flange notch defines on which side of the beam the notches are created.

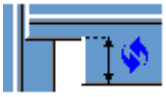
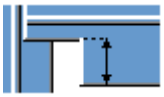
Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

Flange notch shape

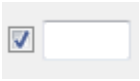
The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

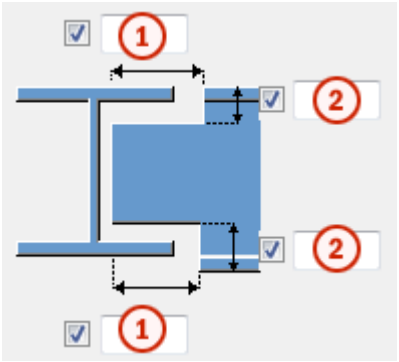
Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.



Cut dimensions



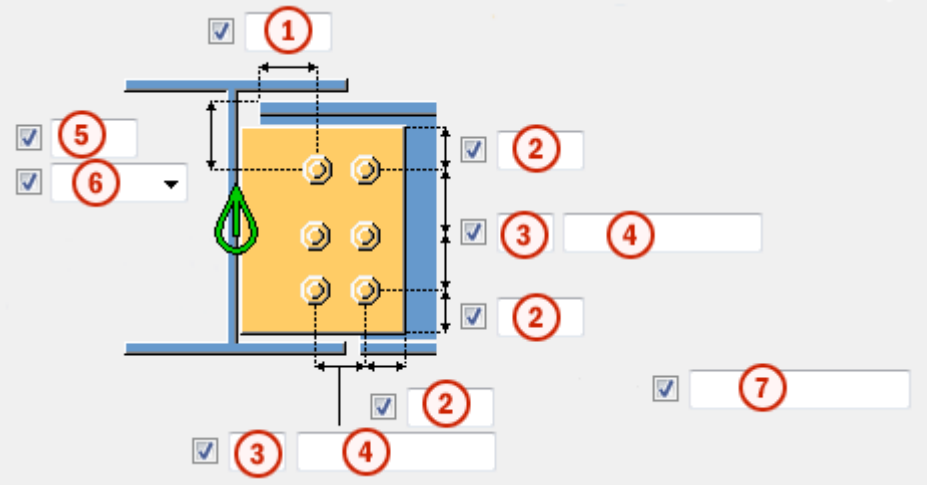
	Description	Default
1	Dimensions for the horizontal flange cuts.	`10 mm`
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest `5 mm`.

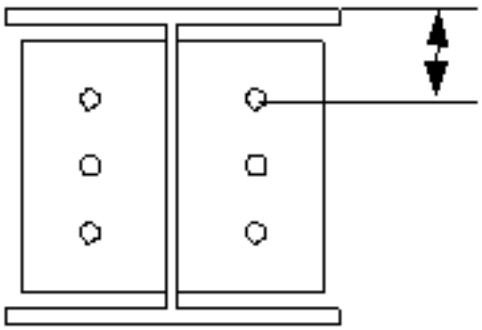
Bolts tab

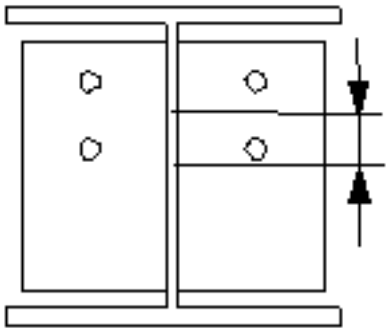
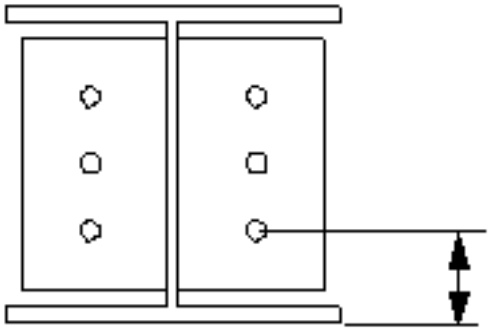
Use the **Bolts** tab to control the properties of the bolts that connect the shear tab to the secondary part.

Bolt group dimensions

Bolt group dimensions affect the size and shape of the shear tab.

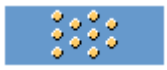


	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position.
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1

Option	Description
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt group orientation

Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Staggered Bolts are staggered in the direction of the secondary part.
	Square Square bolt group is positioned horizontally.
	Sloped Square bolt group is sloped in the direction of the secondary part.

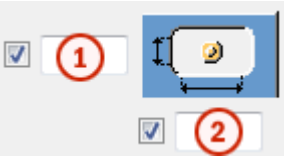
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

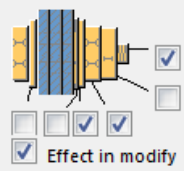


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

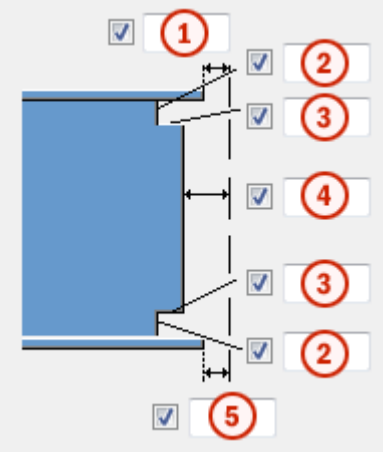
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Beam cut tab

Use the **Beam cut** tab to control the weld access holes, beam end preparations, and flange cuts.

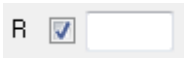
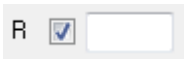
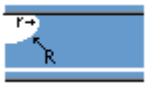
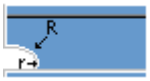
Weld access hole dimensions



	Description
1	Gap between the secondary part top flange and the main part.
2	Vertical dimensions for the top and the bottom weld access holes.
3	Horizontal dimensions for the top and the bottom weld access holes.
4	Gap between the secondary part web and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.

	Description
5	Gap between the secondary part bottom flange and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.

Weld access holes

Option	Option	Description
		Default Round weld access hole AutoDefaults can change this option.
		Round weld access hole
		Square weld access hole
		Round weld access hole with a radius that you can define in 
		Extended cone-shaped weld access hole with a radius and dimensions that you can define in 
		Cone-shaped weld access hole with radiuses that you can define in and Capital **R** defines the large radius (height). Small **r** defines the small radius.

Beam end preparation

Option	Description
	Default Top and bottom flange are prepared. AutoDefaults can change this option.
	Automatic Top and bottom flange are prepared.
	Beam end is not prepared.
	Top and bottom flange are prepared.

Beam end alignment

Option	Description
	Default Beam end is not aligned. Secondary part flange is not aligned with the top and bottom plates. AutoDefaults can change this option.
	Beam end is not aligned.
	Beam end is aligned.

Flange cut

Option for top flange	Option for bottom flange	Description
		Default Flange is not cut. AutoDefaults can change this option.
		Flange is not cut.

Option for top flange	Option for bottom flange	Description
		Flange is cut.

Doubler plates tab

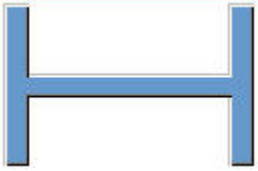
Use the **Doubler plate** tab to create doubler plates to strengthen the web of the main part.

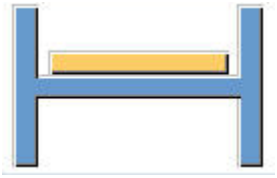
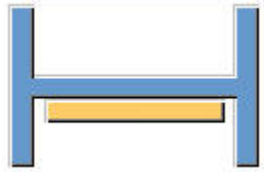
Web plate

Option	Description
Web plate	Web plate thickness and height.

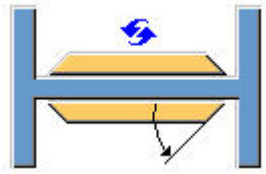
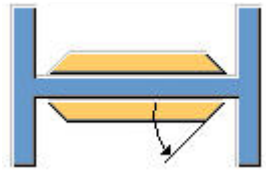
Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Doubler plates

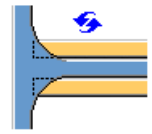
Option	Description
	Default Doubler plates are not created. AutoDefaults can change this option.
	Doubler plates are not created.

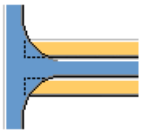
Option	Description
	Doubler plate is created on the far side.
	Doubler plate is created on the near side.
	Doubler plates are created on both sides.

Doubler plate edge shape

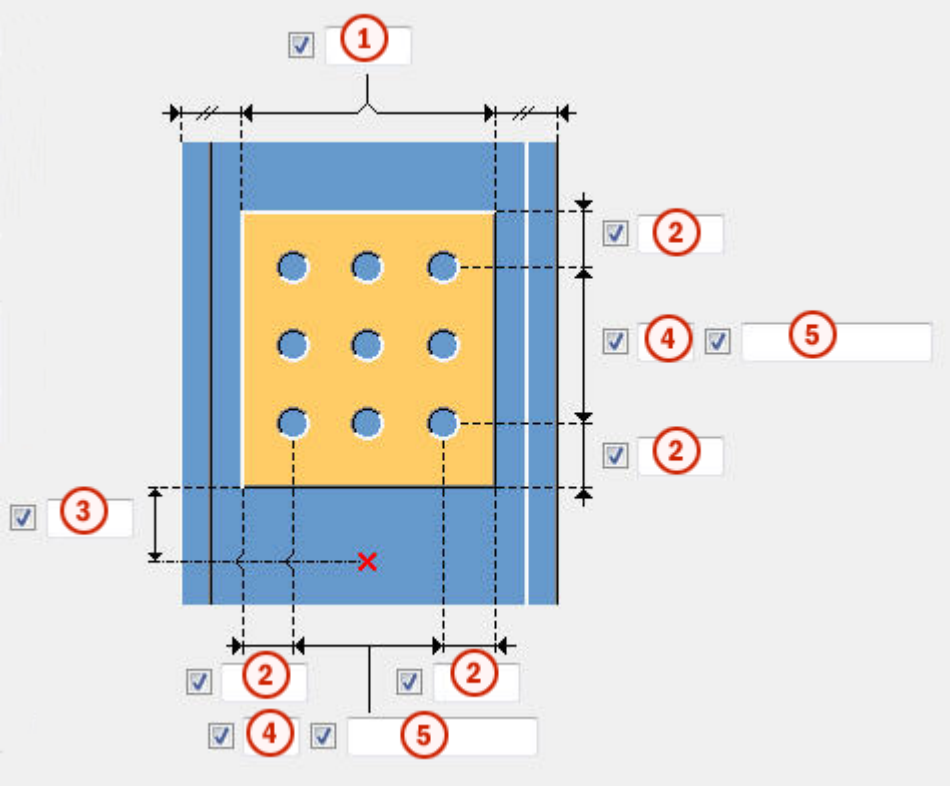
Option	Description
	Default Bevel doubler plates AutoDefaults can change this option.
	Bevel doubler plates Enter the angle in
	Square doubler plates

Doubler plate cuts

Option	Description
	Default Doubler plates are not cut. AutoDefaults can change this option.

Option	Description
	Doubler plates are not cut.
	Doubler plates are cut in the area that connects the main part web and flange.

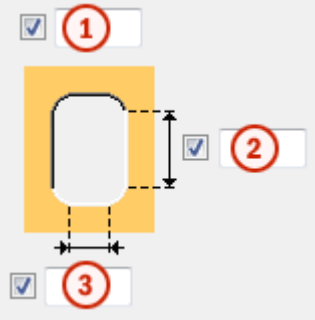
General settings



	Description
1	Edge distance from the column flange.
2	Doubler plate edge distance. Edge distance is the distance from the center of a hole to the edge of the part.
3	Edge distance of the doubler plate in relation to the bottom of the secondary part.
4	Number of holes.

	Description
5	Hole spacing. Use a space to separate hole spacing values. Enter a value for each space between holes. For example, if there are 3 holes, enter 2 values.

Weld hole size



	Description
1	Hole diameter.
2	Slot length.
3	Slot width.

General tab

Click the link below to find out more:

General tab

Design type tab

Click the link below to find out more:

Design type tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

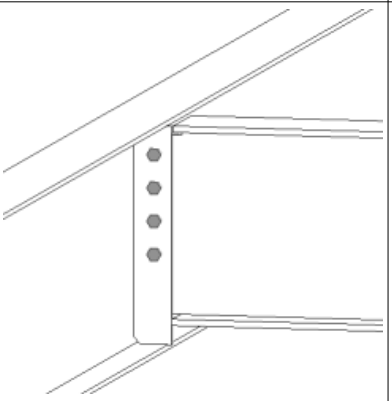
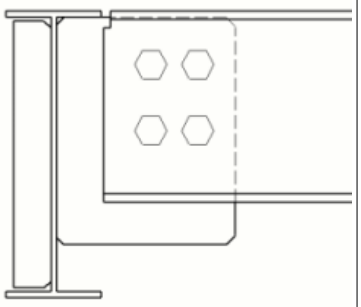
Full depth (184)

Full depth (184) connects two beams with a full depth shear tab. The shear tab is welded to the main beam web and flanges, and bolted to the secondary beam web. The secondary beam can be leveled or sloped. A stiffener plate on the opposite side of the main beam web and the haunch plates welded to the secondary beam flanges are optional.

Objects created

- Shear tab (1 or 2)
- Stiffener (optional)
- Haunch plates (optional)
- Weld backing bars (optional)
- Bolts
- Welds
- Cuts

Use for

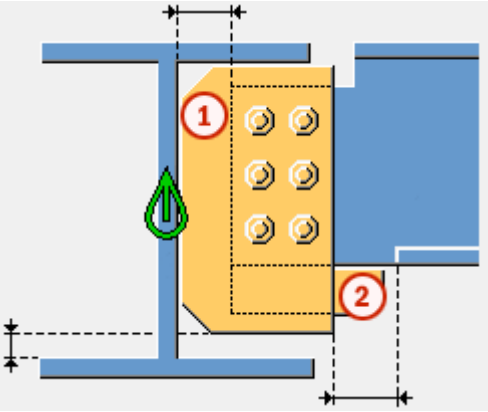
Situation	Description
	Full depth shear tab.
	Full depth connection with a stiffener plate.

Selection order

1. Select the main part (beam).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key



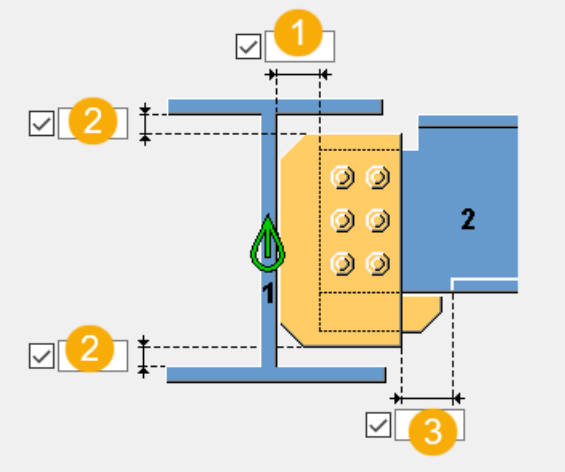
	Part
1	Shear tab
2	Haunch plate

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to control the position of the shear tab, and the beam flange and the web cuts.

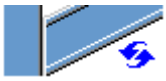
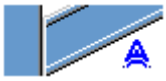
Dimensions



	Description	Default
1	Cut of the secondary part. The cut is defined from the main part web.	20 mm
2	Shear tab edge distance from the main part flange edge.	
3	Size of the strip made to the secondary part flange. The cut of the flange is defined from the shear tab edge.	The flange is automatically stripped when the shear tab crosses the flange. 10 mm

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.
	Square Cuts the end of the secondary beam square.

Option	Description
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.
	Square cut closer to the main part web Cuts the end of the secondary beam square and places the beam closer to the main part web.
	Clipped flange Cuts the corner of the flange at the end of the secondary beam.

Beam web cut

Define how the secondary beam web end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the web bevel when the end of the secondary beam is cut bevel.
	Square Cuts the end of the web square even if the end of the secondary beam is cut bevel.

Beam flange cut

Define how the secondary beam flange end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the flange bevel.

Option	Description
	Square Cuts a part of the flange square and leaves a part of it bevel.

Beam bottom flange cut

Option	Description
	Default Flange cut AutoDefaults can change this option.
	Notch The bottom of the secondary beam is notched if the shear tab crosses the flange. Enter the notch radius and height.
	Flange cut The secondary beam flange is cut on the same side as the shear tab if the shear tab crosses the flange.

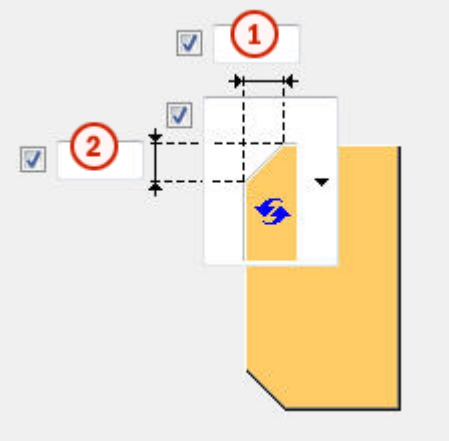
Plates tab

Use the **Plates** tab to control the size, position, number, and shape of the shear tab.

Shear tab plate

Option	Description
Tab plate	Shear tab thickness and width.

Shear tab chamfers



	Description
1	Horizontal dimension of the shear tab chamfer.
2	Vertical dimension of the shear tab chamfer.

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer

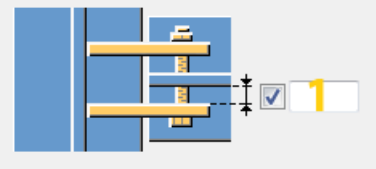
Option	Description
	Concave arc chamfer

Shear tab position

Define the number and the side of shear tabs in single shear tab connections.

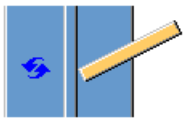
Option	Description
	Default Far side shear tab AutoDefaults can change this option.
	Automatic The component automatically selects either the near side or the far side shear tab. The tab is created to the side of the secondary part when the angle between the main part and the secondary part is less than 90 degrees.
	Far side shear tab
	Near side and far side shear tab
	Near side shear tab

Gap between shear tabs



	Description	Default
1	Gap between the secondary part web and shear tab. This only affects connections with two shear tabs.	0

Shear tab end cut

Option	Description
	Default The shear tab end is not cut. AutoDefaults can change this option.
	Square The shear tab end is not cut.
	Bevel The shear tab end is cut parallel to the main part web.

Stiffeners tab

Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position and type.

Opposite web stiffener plate dimensions

Option	Description
Opposite web stiffener	Opposite web stiffener plate thickness, width and height.

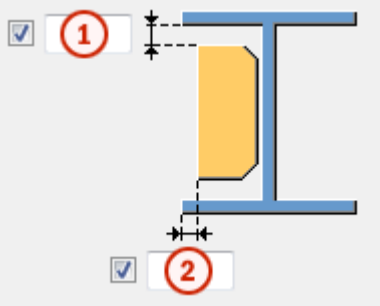
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in

Option	Description	Default
		File menu --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Stiffener creation

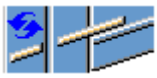
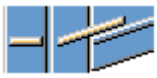
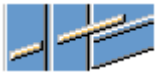
Option	Description
	Default No stiffeners are created. AutoDefaults can change this option.
	Full Creates a full stiffener of the same height as the web of the main part.
	Determined by shear tab Tekla Structures determines the size of the stiffener based on the shear tab size. Tekla Structures attempts to keep the bottom edges of the stiffener plate and shear tab level, if possible.
	Partial Leaves a gap between the stiffener plate and the bottom flange of the main part.
	No stiffeners are created.

Stiffener gap

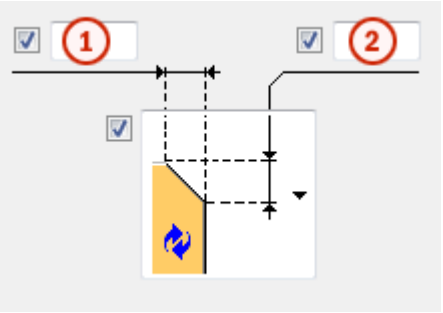


	Description
1	Size of the gap between the main part flange and the stiffener.
2	Distance from the edge of the main part flange to the edge of the stiffener.

Stiffener orientation

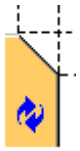
Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are perpendicular to the main part.
	Stiffeners are parallel to the secondary part.

Chamfer dimensions



	Description
1	Horizontal dimension of the chamfer.
2	Vertical dimension of the chamfer.

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Haunch tab

Use the **Haunch** tab to control the haunch plate creation and chamfers in the secondary beam flanges.

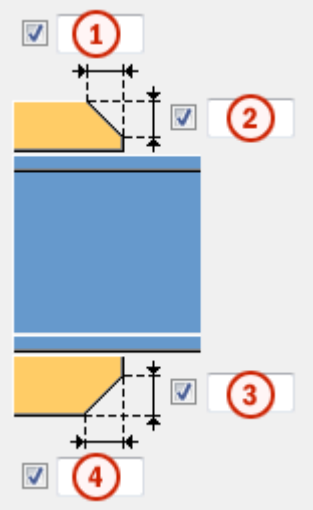
Haunch plates

Option	Description
Top plate	Top haunch plate thickness, width and height.
Bottom plate	Bottom haunch plate thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

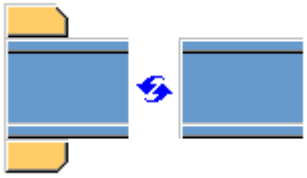
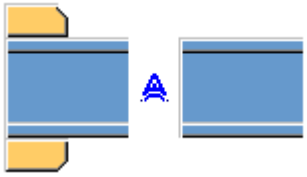
Option	Description	Default
	assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Haunch plate chamfers



	Description
1	Width of the top haunch plate chamfer.
2	Height of the top haunch plate chamfer.
3	Height of the bottom haunch plate chamfer.
4	Width of the bottom haunch plate chamfer.

Haunch plate creation

Option	Description
	Default Top and bottom haunch plates are created, if needed. AutoDefaults can change this option.
	Automatic Top or bottom haunch plate or both are created, if needed.
	Top and bottom haunch plates are created. To create a single plate, enter `0` in the thickness (**t**) field for the plate you do not need (top or bottom plate).
	Haunch plates are not created.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

Automatic notching options affect both the top and the bottom flange.

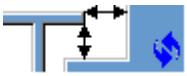
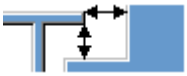
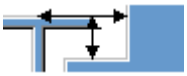
Notch shape

Automatic notching is switched on when you select a notch shape.

Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.

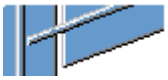
Option	Description
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

Enter the horizontal and vertical values for the cuts.

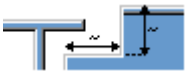
Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.

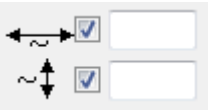
Option	Description
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.



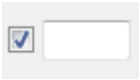
Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.



Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

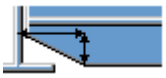
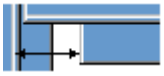
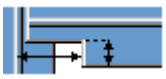
Side of flange notch

The side of flange notch defines on which side of the beam the notches are created.

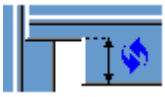
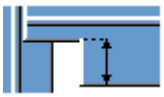
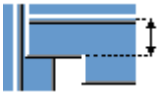
Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

Flange notch shape

The flange notch shape defines the notch shape in the beam flange.

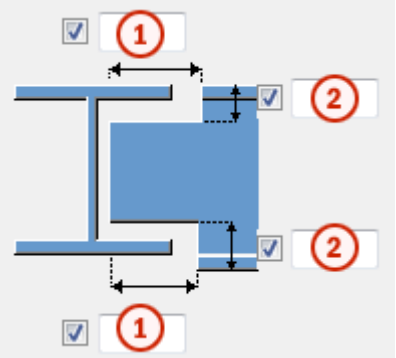
Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

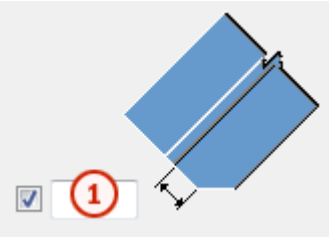
Enter the value for flange notch depth.

Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	`10 mm`
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest `5 mm`.

Dimension from web to flange cut



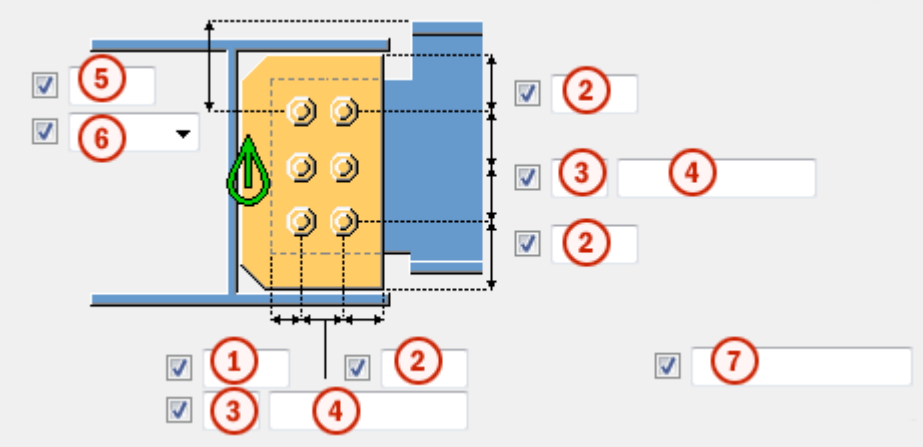
	Description
1	Define the distance between the web and the flange cut.

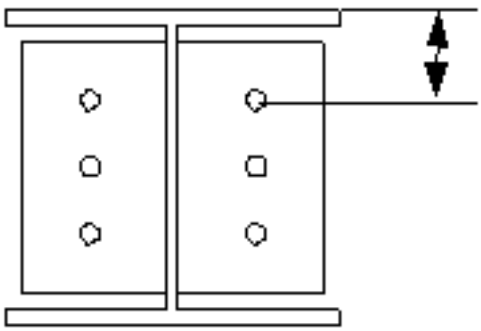
Bolts tab

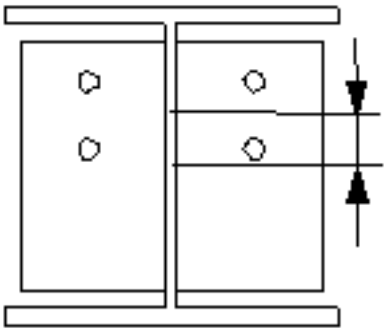
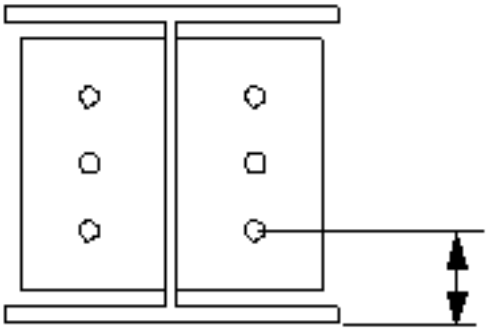
Use the **Bolts** tab to control the properties of the bolts that connect the shear tab to the secondary part.

Bolt group dimensions

Bolt group dimensions affect the size and shape of the shear tab.

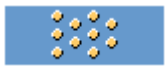


	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position.
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1

Option	Description
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt group orientation

Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Staggered Bolts are staggered in the direction of the secondary part.
	Square Square bolt group is positioned horizontally.
	Sloped Square bolt group is sloped in the direction of the secondary part.

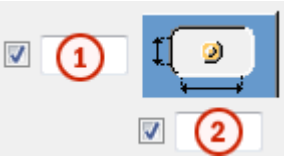
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

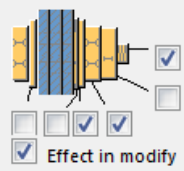


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Beam cut tab

Use the **Beam cut** tab to control weld backing bars, weld access holes, beam end preparations, and flange cuts.

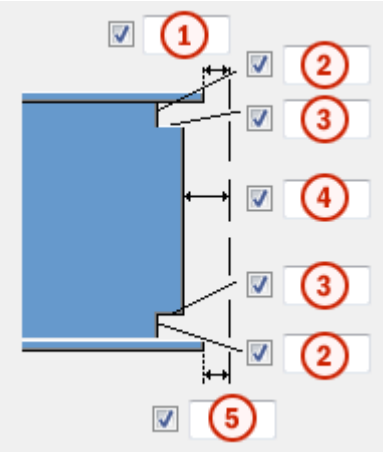
Weld backing bar

Option	Description
Weld backing bar	Weld backing bar thickness and width.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Option	Description	Default
Finish	Describes how the part surface has been treated.	

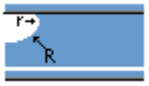
Weld access hole dimensions



	Description
1	Gap between the secondary part top flange and the main part.
2	Vertical dimensions for the top and the bottom weld access holes.
3	Horizontal dimensions for the top and the bottom weld access holes.
4	Gap between the secondary part web and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.
5	Gap between the secondary part bottom flange and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.

Weld access holes

Option	Description	Default
	Default Round weld access hole AutoDefaults can change this option.	
	Round weld access hole	

Option	Description	Default
	Square weld access hole	
	Diagonal weld access hole	
	Round weld access hole with a radius that you can define in r ☑	
	Extended cone-shaped weld access hole with a radius and dimensions that you can define in R ☑ and Top Prep x ☑ Bottom Prep x ☑	
	Cone-shaped weld access hole with radiuses that you can define in R ☑ and r ☑ Capital **R** defines the large radius (height). Small **r** defines the small radius.	R = 35 r = 10

Beam end preparation

Option	Description
	Default Top and bottom flange are prepared. AutoDefaults can change this option.
	Automatic Top and bottom flange are prepared.

Option	Description
	Beam end is not prepared.
	Top and bottom flange are prepared.
	Top flange is prepared.
	Bottom flange is prepared.

Flange cut

Option for top flange	Option for bottom flange	Description
		Default Flange is not cut. AutoDefaults can change this option.
		Flange is not cut.
		Flange is cut.

Weld backing bars

Option for top backing bar	Option for bottom backing bar	Description
		Default Backing bars are created inside the flanges. AutoDefaults can change this option.
		No backing bars are created.
		Backing bars are created inside the flanges.

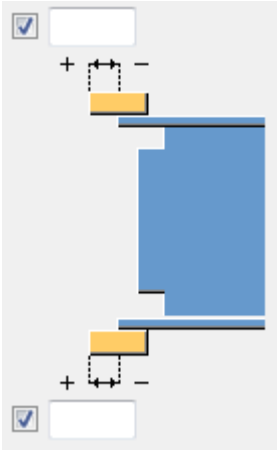
Option for top backing bar	Option for bottom backing bar	Description
		Backing bars are created outside the flanges.

Weld backing bar length

Enter the length of the weld backing bar in the box below the options.

Option	Description
	Default Absolute length of the backing bar AutoDefaults can change this option.
	Absolute length of the backing bar
	Extension beyond the edge of the flange

Weld backing bar position

Option	Description
	Enter a positive or a negative value to move the front end of the backing bar relative to the end of the flange.

Assembly type

Define the location where the weld backing bar welds are made. When you select the **Workshop** option, Tekla Structures includes the backing bars in the assembly.

General tab

Click the links below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds tab

Click the link below to find out more:

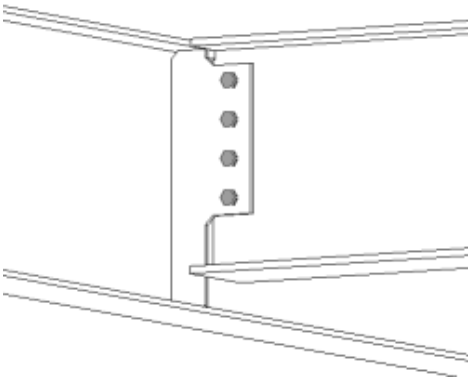
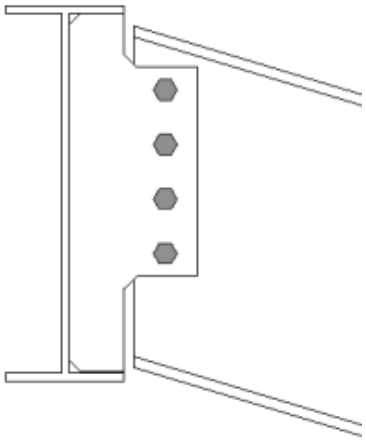
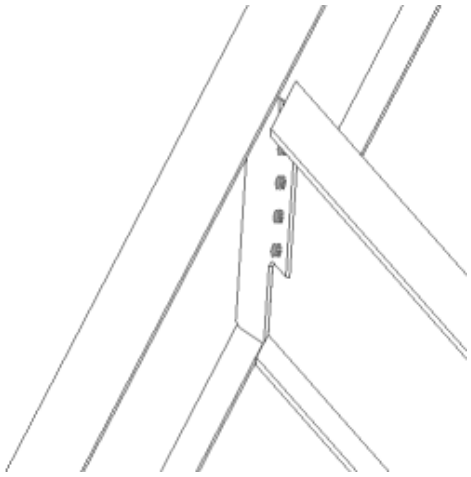
Full depth S (185)

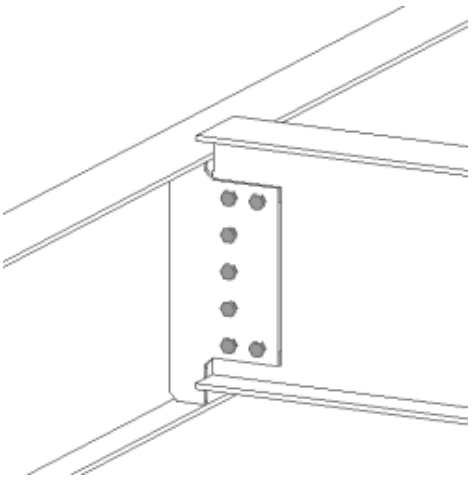
Full depth S (185) connects two beams with a full depth shear tab. The shear tab is welded to the main beam web and flanges, and bolted to the secondary beam web. The secondary beam can be leveled or sloped. A stiffener plate on the opposite side of the main beam web and the haunch plates welded to the secondary beam flanges are optional.

Objects created

- Shear tabs (1 or 2)
- Stiffener (optional)
- Haunch plates (optional)
- Weld backing bars (optional)
- Bolts
- Welds
- Cuts

Use for

Situation	Description
	Full depth shear tab.
	Full depth shear tab. The secondary part is sloped.
	Full depth shear tab. The secondary part is sloped and skewed.

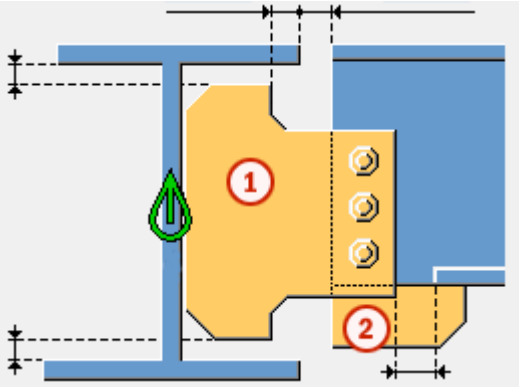
Situation	Description
	Full depth shear tab. The secondary part is offset. Some bolts have been deleted.

Selection order

1. Select the main part (beam).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key



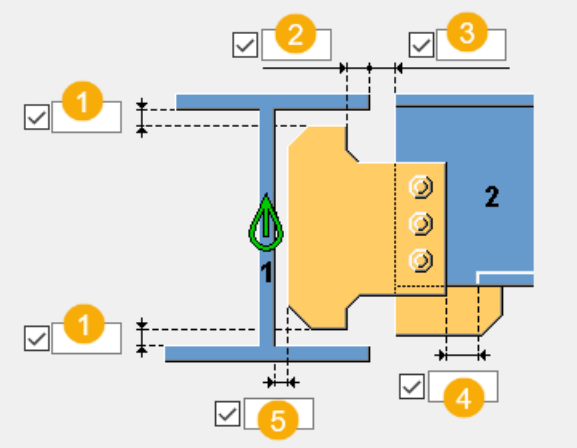
	Part
1	Shear tab
2	Haunch plate

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to control the position of the shear tab, and the beam flange and web cuts.

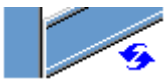
Dimensions

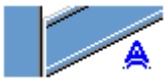


	Description	Default
1	Shear tab edge distance from the main part flange edge.	0
2	Shear tab edge distance from the main part flange edge.	0
3	Cut of the secondary part. Cutting the secondary part creates a gap between the main part and the secondary part.	20 mm
4	Size of the strip made to the secondary part flange. The cut of the flange is defined from the shear tab edge.	The flange is automatically stripped when the shear tab crosses the flange. 20 mm
5	Gap between the shear tab and the main part web.	

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.

Option	Description
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.
	Square Cuts the end of the secondary beam square.
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.
	Square cut closer to the main part web Cuts the end of the secondary beam square and places the beam closer to the main part web.
	Clipped flange Cuts the corner of the flange at the end of the secondary beam.

Beam web cut

Define how the secondary beam web end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the web bevel when the end of the secondary beam is cut bevel.
	Square Cuts the end of the web square even if the end of the secondary beam is cut bevel.

Beam flange cut

Define how the secondary beam flange end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the flange bevel.
	Square Cuts a part of the flange square and leaves a part of it bevel.

Beam bottom flange cut

Option	Description
	Default Flange cut AutoDefaults can change this option.
	Notch The bottom of the secondary beam is notched if the shear tab crosses the flange. Enter the notch radius and height.
	Flange cut The secondary beam flange is cut on the same side as the shear tab if the shear tab crosses the flange.

Plates tab

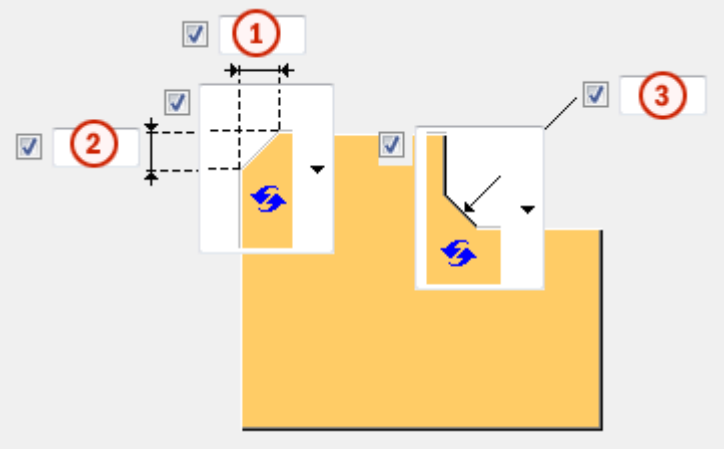
Use the **Plates** tab to control the size, position, number, and shape of the shear tab.

Shear tab plate

Option	Description
Tab plate	Shear tab plate thickness, width and height.

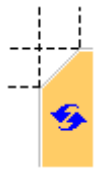
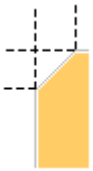
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Shear tab chamfers



	Description
1	Horizontal dimension of the shear tab chamfer.
2	Vertical dimension of the shear tab chamfer.
3	Vertical and the horizontal dimension of the shear tab chamfer.

Chamfer type

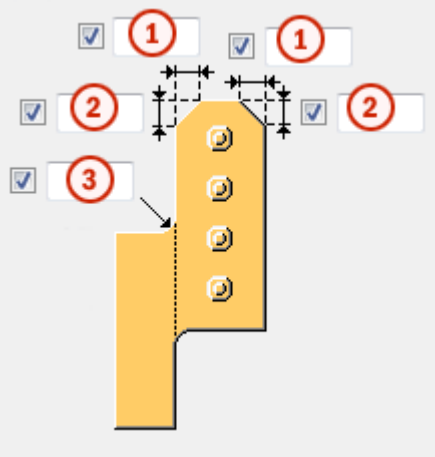
Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Chamfer type dimensions

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer

Option	Description
	Line chamfer
	Concave arc chamfer

Inner shear tab chamfers



	Description
1	Horizontal dimension of the shear tab chamfer.
2	Vertical dimension of the shear tab chamfer.
3	Radius and the vertical dimensions of the shear tab inner chamfer.

Chamfer type

Option	Option	Description
		Default No chamfer AutoDefaults can change this option.
		No chamfer
		Line chamfer

Option	Option	Description
		Convex arc chamfer
		Concave arc chamfer

Inner chamfer type

Option	Description
	Default Concave arc chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Concave arc chamfer
	Convex arc chamfer

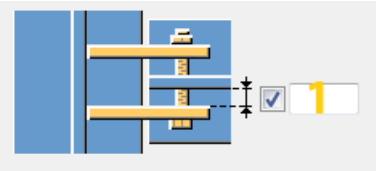
Shear tab position

Define the number and the side of shear tabs in single shear tab connections.

Option	Description
	Default Far side shear tab AutoDefaults can change this option.
	Automatic The component automatically selects either the near side or the far side shear tab. The tab is created to the side of the secondary part when the angle between the main part and the secondary part is less than 90 degrees.

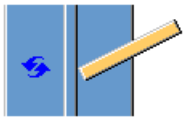
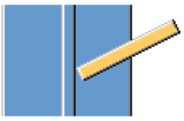
Option	Description
	Far side shear tab
	Near side and far side shear tab
	Near side shear tab

Gap between shear tabs



	Description	Default
1	Gap between the secondary part web and shear tab. This only affects connections with two shear tabs.	0

Shear tab end cut

Option	Description
	Default The shear tab end is not cut. AutoDefaults can change this option.
	Square The shear tab end is not cut.
	Bevel The shear tab end is cut parallel to the main part web.

Shear tab orientation

Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Sloped The shear tab is sloped in the direction of the secondary beam. Both vertical edges of the shear tab are cut parallel to the end of the secondary beam.
	Square

Stiffeners tab

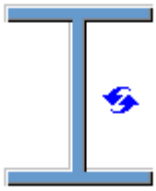
Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position and type.

Opposite web stiffener plate dimensions

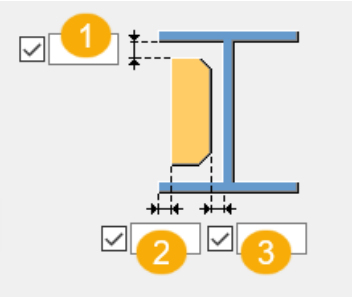
Option	Description
Opposite web stiffener	Opposite web stiffener plate thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Stiffener creation

Option	Description
	Default No stiffeners are created. AutoDefaults can change this option.
	Full Creates a full stiffener of the same height as the web of the main part.
	Determined by shear tab Tekla Structures determines the size of the stiffener based on the shear tab size. Tekla Structures attempts to keep the bottom edges of the stiffener plate and shear tab level, if possible.
	Partial Leaves a gap between the stiffener plate and the bottom flange of the main part.
	No stiffeners are created.

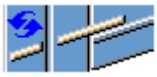
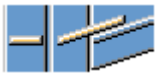
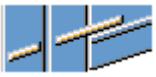
Stiffener gap



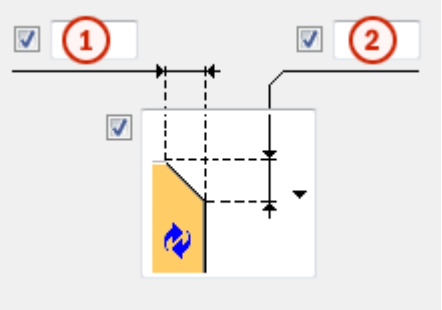
	Description
1	Size of the gap between the main part flange and the stiffener.

	Description
2	Distance from the edge of the main part flange to the edge of the stiffener.
3	Gap between the opposite web stiffener and main part web.

Stiffener orientation

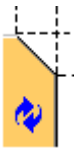
Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are perpendicular to the main part.
	Stiffeners are parallel to the secondary part.

Chamfer dimensions



	Description
1	Horizontal dimension of the chamfer.
2	Vertical dimension of the chamfer.

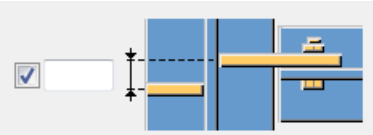
Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer

Option	Description
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Opposite web stiffener offset

Define the opposite web stiffener offset from the shear tab center line.



Haunch tab

Use the **Haunch** tab to control the haunch plate creation and chamfers in the secondary beam flanges.

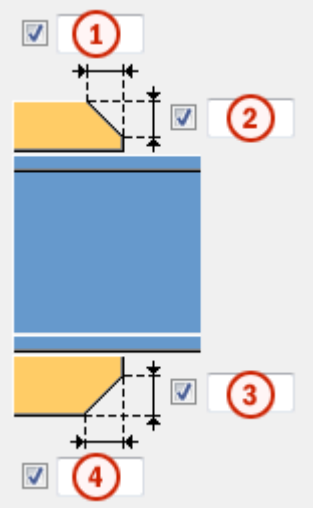
Haunch plates

Option	Description
Top plate	Top haunch plate thickness, width and height.
Bottom plate	Bottom haunch plate thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.

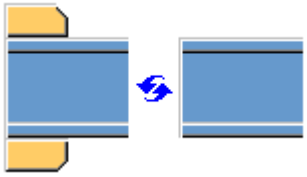
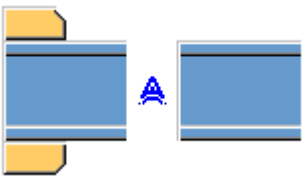
Option	Description	Default
Name	Name that is shown in drawings and reports.	

Haunch plate chamfers



	Description
1	Width of the top haunch plate chamfer.
2	Height of the top haunch plate chamfer.
3	Height of the bottom haunch plate chamfer.
4	Width of the bottom haunch plate chamfer.

Haunch plate creation

Option	Description
	Default Top and bottom haunch plates are created, if needed. AutoDefaults can change this option.
	Automatic Top or bottom haunch plate or both are created, if needed.

Option	Description
	Top and bottom haunch plates are created. To create a single plate, enter `0` in the thickness (**t**) field for the plate you do not need (top or bottom plate).
	Haunch plates are not created.

Notch tab

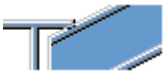
Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

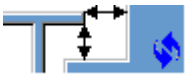
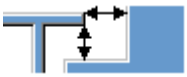
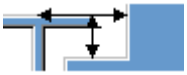
Automatic notching options affect both the top and the bottom flange.

Notch shape

Automatic notching is switched on when you select a notch shape.

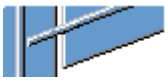
Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

Enter the horizontal and vertical values for the cuts.

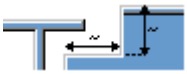
Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.

Option	Description
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.

Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.

Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

Side of flange notch

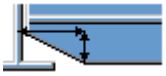
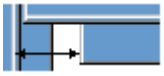
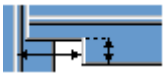
The side of flange notch defines on which side of the beam the notches are created.

Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

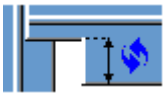
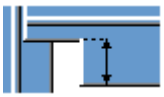
Flange notch shape

The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.

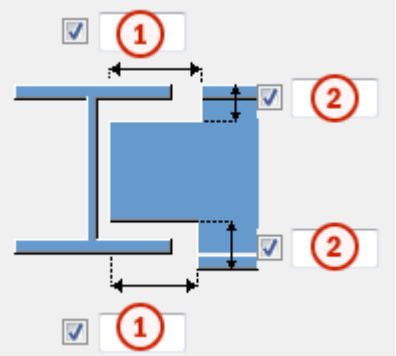
Option	Description
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

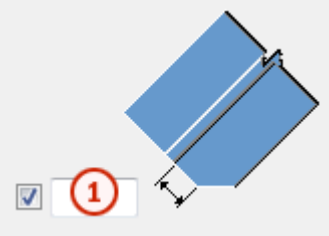
Enter the value for flange notch depth.

Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	10 mm
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest 5 mm.

Dimension from web to flange cut



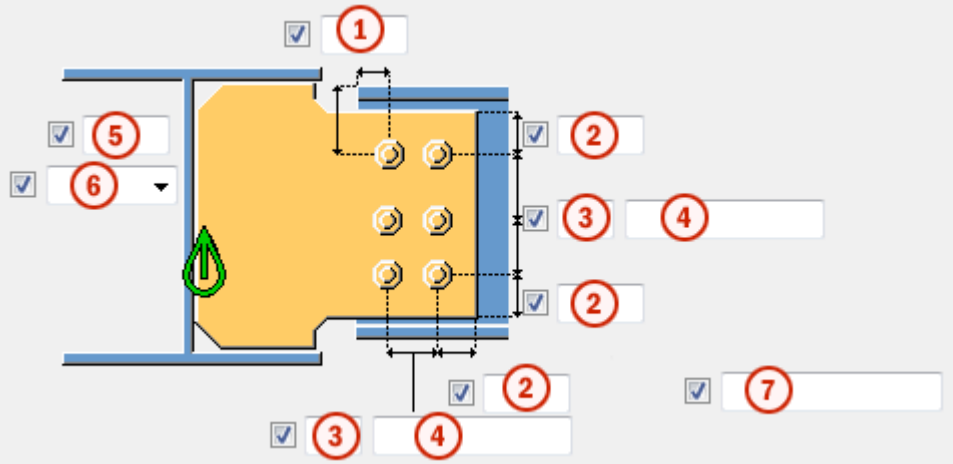
	Description
1	Define the distance between the web and the flange cut.

Bolts tab

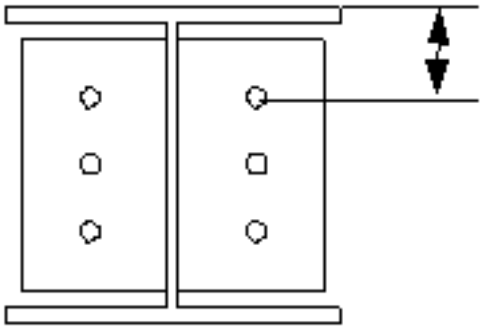
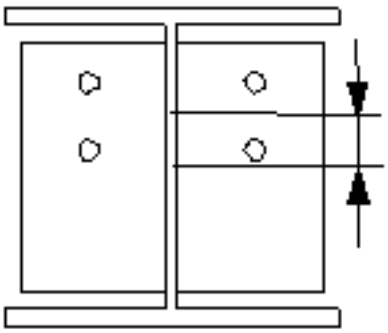
Use the **Bolts** tab to control the properties of the bolts that connect the shear tab to the secondary part.

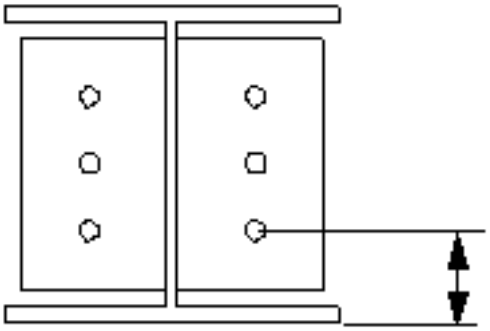
Bolt group dimensions

Bolt group dimensions affect the size and shape of the shear tab.



	Description
1	Dimension for horizontal bolt group position.

	Description
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position.
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part. 

	Description
	• **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt group orientation

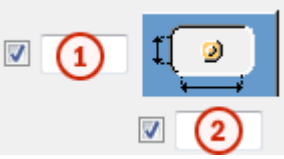
Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Staggered Bolts are staggered in the direction of the secondary part.
	Square Square bolt group is positioned horizontally.
	Sloped Square bolt group is sloped in the direction of the secondary part.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

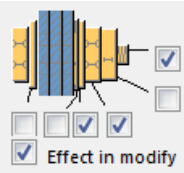


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Bolting direction

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1
	Bolting direction 2

Beam cut tab

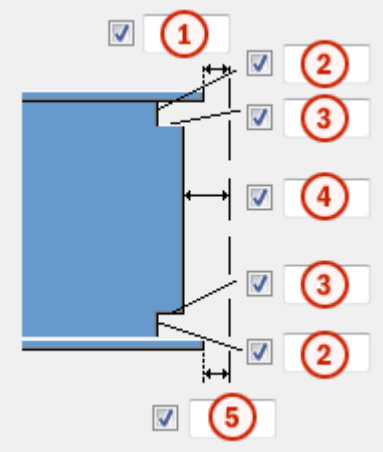
Use the **Beam cut** tab to control weld backing bars, weld access holes, beam end preparations, and flange cuts.

Weld backing bar

Option	Description
Weld backing bar	Weld backing bar thickness and width.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

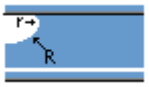
Weld access hole dimensions



	Description
1	Gap between the secondary part top flange and the main part.
2	Vertical dimensions for the top and the bottom weld access holes.
3	Horizontal dimensions for the top and the bottom weld access holes.
4	Gap between the secondary part web and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.
5	Gap between the secondary part bottom flange and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.

Weld access holes

Option	Description	Default
	Default Round weld access hole AutoDefaults can change this option.	
	Round weld access hole	
	Square weld access hole	
	Diagonal weld access hole	

Option	Description	Default
	Round weld access hole with a radius that you can define in r ☑	
	Extended cone-shaped weld access hole with a radius and dimensions that you can define in R ☑ and Top Prep x ☑ Bottom Prep x ☑	
	Cone-shaped weld access hole with radiuses that you can define in R ☑ and r ☑ Capital **R** defines the large radius (height). Small **r** defines the small radius.	R = 35 r = 10

Beam end preparation

Option	Description
	Default Top and bottom flange are prepared. AutoDefaults can change this option.
	Automatic Top and bottom flange are prepared.
	Beam end is not prepared.
	Top and bottom flange are prepared.

Option	Description
	Top flange is prepared.
	Bottom flange is prepared.

Flange cut

Option for top flange	Option for bottom flange	Description
		Default Flange is not cut. AutoDefaults can change this option.
		Flange is not cut.
		Flange is cut.

Weld backing bars

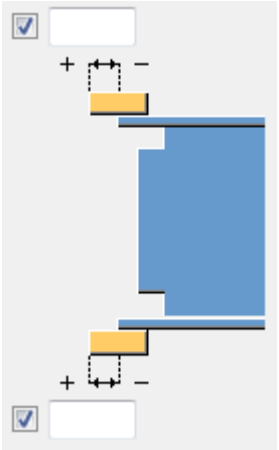
Option for top backing bar	Option for bottom backing bar	Description
		Default Backing bars are created inside the flanges. AutoDefaults can change this option.
		No backing bars are created.
		Backing bars are created inside the flanges.
		Backing bars are created outside the flanges.

Weld backing bar length

Enter the length of the weld backing bar in the box below the options.

Option	Description
	Default Absolute length of the backing bar AutoDefaults can change this option.
	Absolute length of the backing bar
	Extension beyond the edge of the flange

Weld backing bar position

Option	Description
	Enter a positive or a negative value to move the front end of the backing bar relative to the end of the flange.

Assembly type

Define the location where the weld backing bar welds are made. When you select the **Workshop** option, Tekla Structures includes the backing bars in the assembly.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

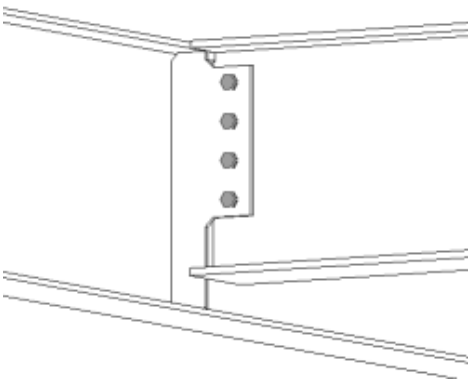
JP Full depth special (185)

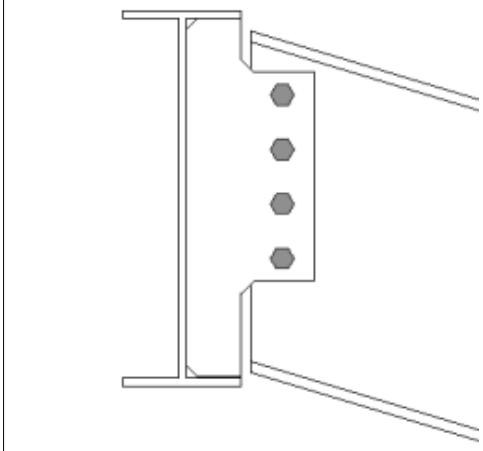
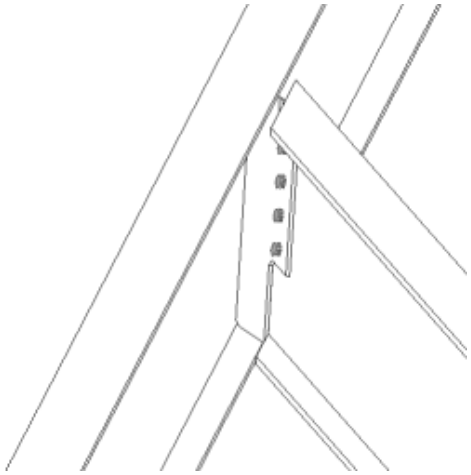
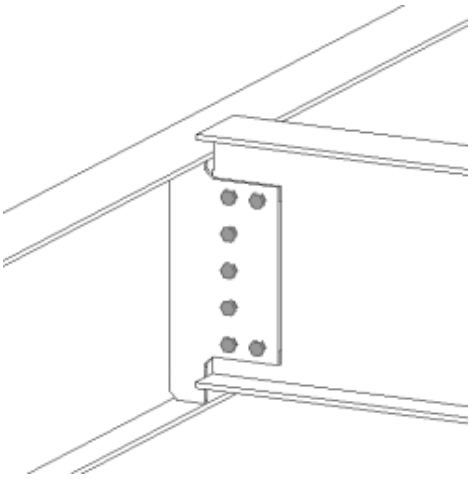
JP Full depth special (185) connects two beams with a full depth shear tab. The shear tab is welded to the main beam web and flanges, and bolted to the secondary beam web. The secondary beam can be leveled or sloped. A stiffener plate can be created on the opposite side of the main beam web.

Objects created

- Shear tabs (1 or 2)
- Stiffener (optional)
- Bolts
- Welds

Use for

Situation	Description
	Full depth shear tab.

Situation	Description
	Full depth shear tab. The secondary part is sloped.
	Full depth shear tab. The secondary part is sloped and skewed.
	Full depth shear tab. The secondary part is offset. Some bolts have been deleted.

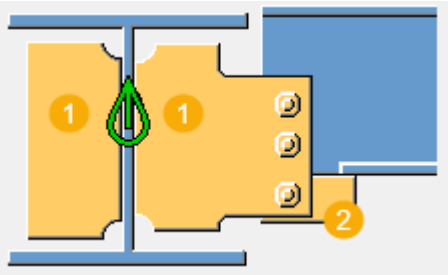
Selection order

1. Select the main part (beam).

2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

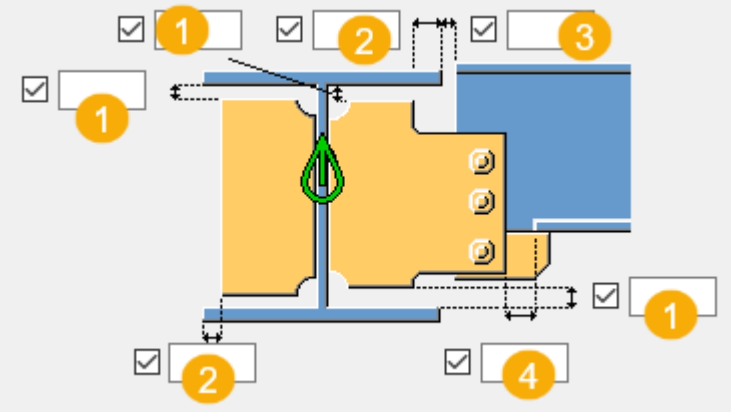


	Description
1	Shear tab
2	Stiffener

Picture tab

Use the **Picture** tab to control the position of the shear tab, and the beam flange and web cuts.

Dimensions

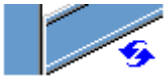
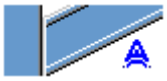


1	Shear tab edge distance from the main part flange edge.
2	Shear tab edge distance from the main part flange edge.
3	Cut of the secondary part. Cutting the secondary part creates a gap between the main part and the secondary part.

4	Size of the strip made to the secondary part flange. The cut of the flange is defined from the shear tab edge.

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.
	Square Cuts the end of the secondary beam square.
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.
	Square cut closer to the main part web Cuts the end of the secondary beam square and places the beam closer to the main part web.
	Clipped flange Cuts the corner of the flange at the end of the secondary beam.

Beam web cut

Define how the secondary beam web end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.

Option	Description
	Bevel Cuts the end of the web bevel when the end of the secondary beam is cut bevel.
	Square Cuts the end of the web square even if the end of the secondary beam is cut bevel.

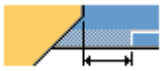
Beam flange cut

Define how the secondary beam flange end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the flange bevel.
	Square Cuts a part of the flange square and leaves a part of it bevel.

Beam bottom flange cut

Option	Description
	Default Flange cut AutoDefaults can change this option.
	Notch The bottom of the secondary beam is notched if the shear tab crosses the flange. Enter the notch radius and height.
	Flange cut The secondary beam flange is cut on the same side as the shear tab if the shear tab crosses the flange.

Option	Description
	Shear tab cut Enter the dimension to the secondary part flange cut.

Shear tab shape

Option	Description
	Default Square cut AutoDefaults can change this option.
	Square cut
	Bevel cut

Plates tab

Use the **Plates** tab to control the size, position, number, and shape of the shear tab.

Shear tab plate

Option	Description
Tab plate	Shear tab plate thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
Name	Name that is shown in drawings and reports.	

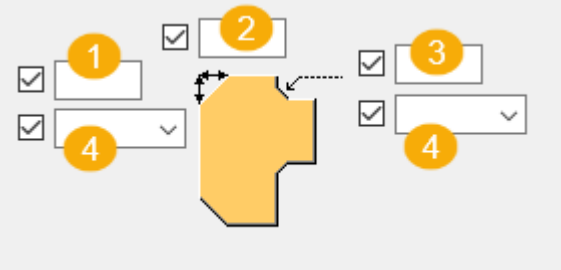
Shear tab position

Option	Description
	Default Near side shear tab AutoDefaults can change this option.
	Automatic The component automatically selects either the near side or the far side shear tab. The tab is created to the side of the secondary part when the angle between the main part and the secondary part is less than 90 degrees.
	Near side shear tab
	Near side and far side shear tab
	Far side shear tab

Shear tab orientation

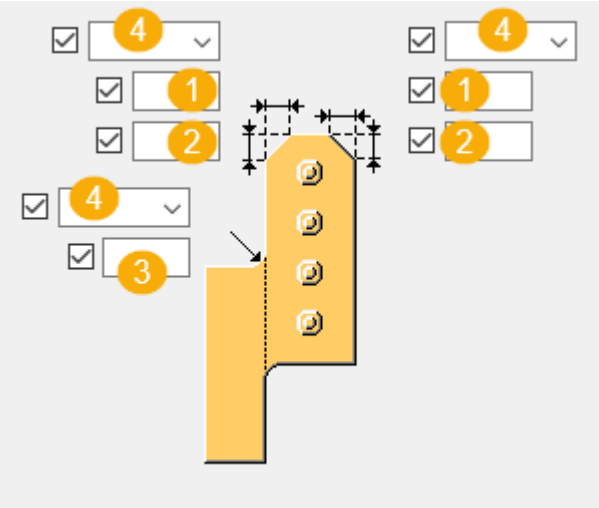
Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Sloped The shear tab is sloped in the direction of the secondary beam. Both vertical edges of the shear tab are cut parallel to the end of the secondary beam.
	Square

Shear tab chamfers



	Description
1	Vertical dimension of the shear tab chamfer.
2	Horizontal dimension of the shear tab chamfer.
3	Shear tab chamfer radius.
4	Select the chamfer type.

Inner shear tab chamfers



	Description
1	Horizontal dimension of the shear tab chamfer.
2	Vertical dimension of the shear tab chamfer.
3	Shear tab chamfer radius.
4	Select the chamfer type.

Stiffeners tab

Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position and type.

Opposite web stiffener plate dimensions

Option	Description
Opposite web stiffener	Opposite web stiffener plate thickness, width and height.

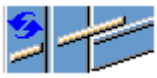
Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Stiffener creation

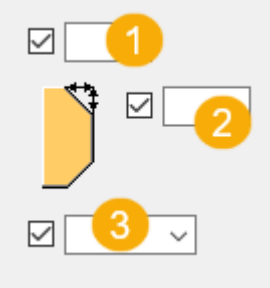
Option	Description
	Default No stiffeners are created. AutoDefaults can change this option.
	Full Creates a full stiffener of the same height as the web of the main part.
	Determined by shear tab Tekla Structures determines the size of the stiffener based on the shear tab size. Tekla Structures attempts to keep the bottom edges of the stiffener plate and shear tab level, if possible.

Option	Description
	Partial Leaves a gap between the stiffener plate and the bottom flange of the main part.
	No stiffeners are created.

Stiffener orientation

Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are perpendicular to the main part.
	Stiffeners are parallel to the secondary part.

Chamfer dimensions

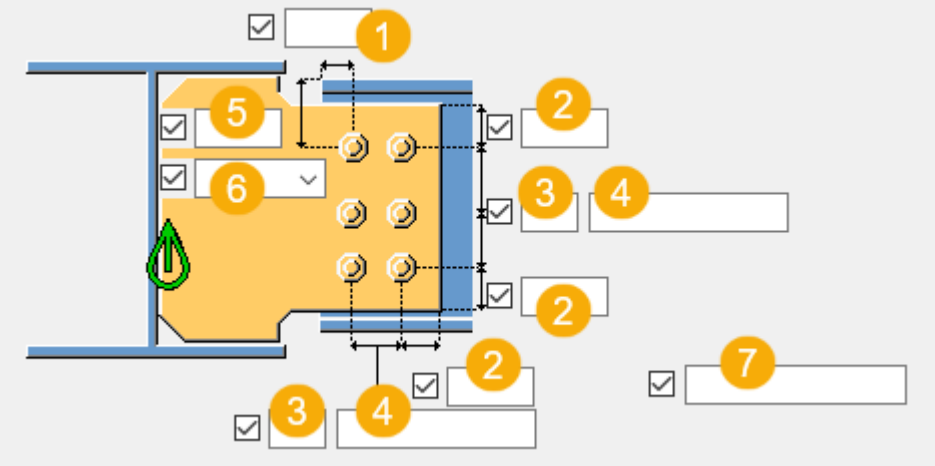


	Description
1	Horizontal dimension of the chamfer.
2	Vertical dimension of the chamfer.
3	Select the chamfer type.

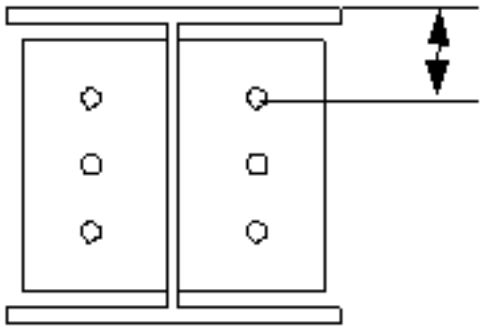
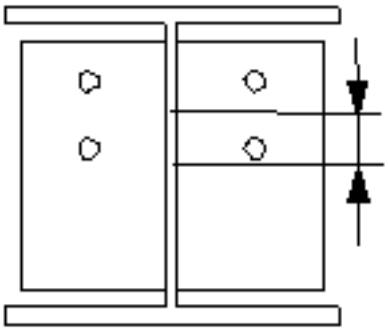
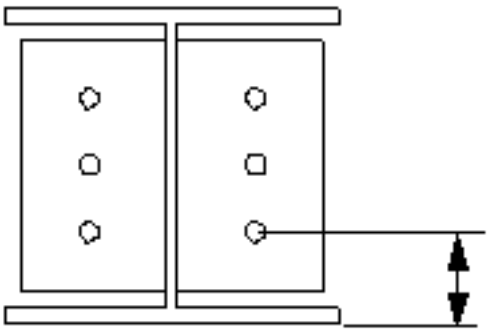
Bolts tab

Use the **Bolts** tab to control the properties of the bolts that connect the shear tab to the secondary part.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position.

	Description
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

	Description
7	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt group orientation

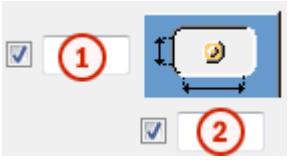
Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Staggered Bolts are staggered in the direction of the secondary part.
	Square Square bolt group is positioned horizontally.
	Sloped Square bolt group is sloped in the direction of the secondary part.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



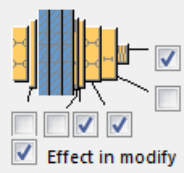
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	

Option	Description	Default
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

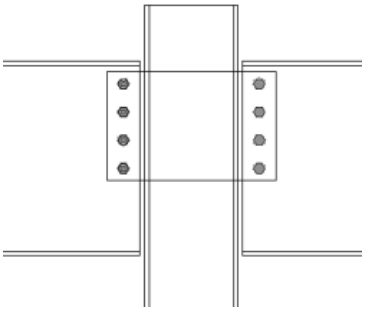
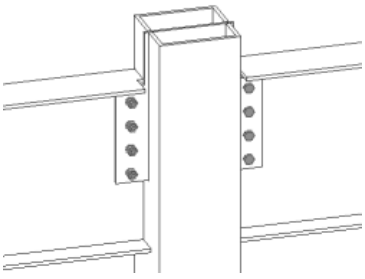
Shear plate tube column (189)

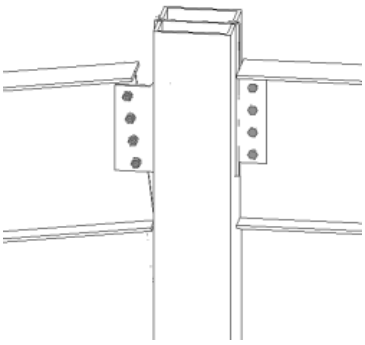
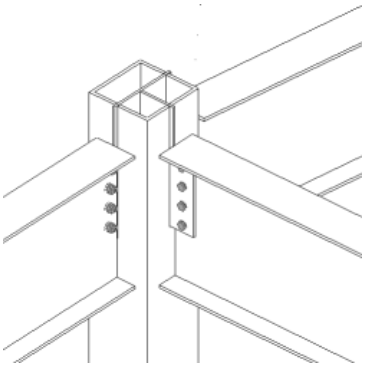
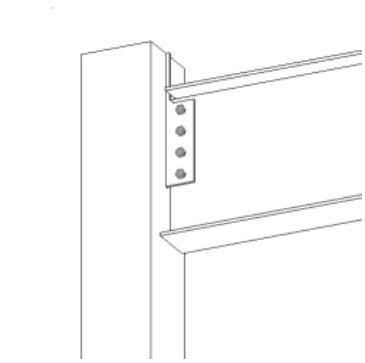
Shear plate tube column (189) connects a beam to a tube column with a shear tab. The shear tab goes through the tube column.

Objects created

- Shear tab (1 or 2)
- Bolts
- Welds
- Cuts

Use for

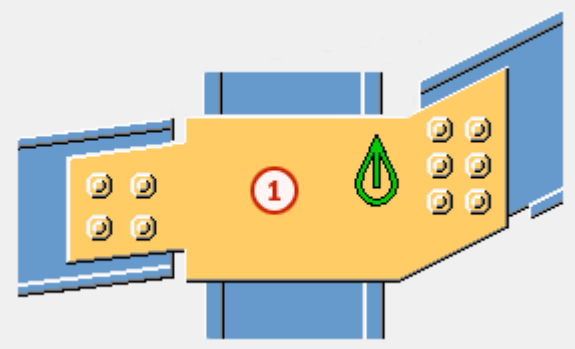
Situation	Description
	Shear tab through a tube column with two secondary parts.
	Shear tab through a tube column with two secondary parts. Shear tab extends to the top of the column.

Situation	Description
	Shear tab through a tube column with two secondary parts and bolt alignment options. The secondary parts can be leveled and/or sloped.
	Shear tab through a tube column with two secondary parts. A third secondary part has been added after the connection was created.
	Shear tab through a tube column with one secondary part.

Selection order

1. Select the main part (column).
2. Select the first secondary part (beam).
3. Select the second secondary part (beam).
4. Click the middle mouse button to create the component.

Part identification key



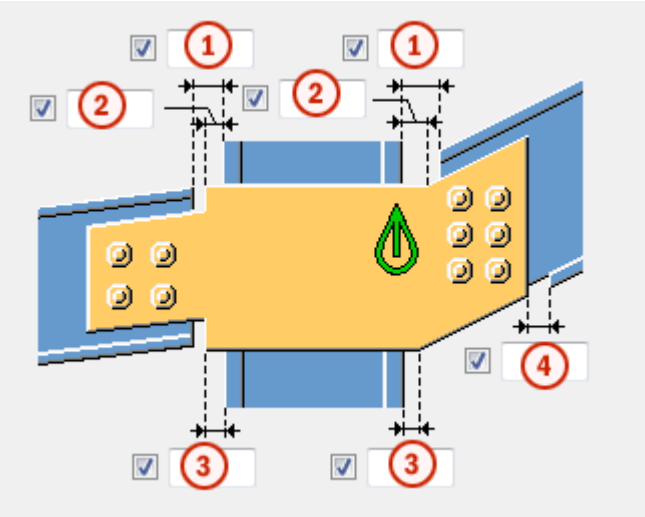
	Part
1	Shear tab

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to control the position of the shear tab and to define how the beam ends are cut.

Dimensions

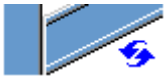
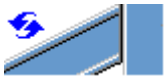
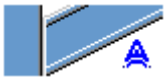
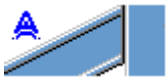
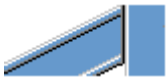


	Description	Default
1	Cut of the secondary part. Cutting the secondary part creates a gap between the main part and the secondary part.	20 mm

	Description	Default
2	Distance from the edge of the main part to the top corner of the shear tab.	5 mm
3	Distance from the edge of the main part to the bottom corner of the shear tab.	5 mm
4	Size of the strip made to the secondary part flange. The cut of the flange is defined from the shear tab edge.	The flange is automatically stripped when the shear tab crosses the flange. 20 mm

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Option	Description
		Default Bevel AutoDefaults can change this option.
		Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.
		Square Cuts the end of the secondary beam square.
		Bevel Cuts the end of the secondary beam parallel to the edge of the main part.

Plates tab

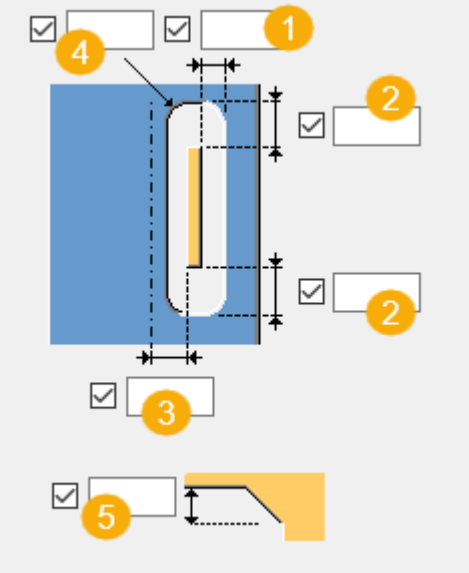
Use the **Plates** tab to control the size, position, number, orientation and shape of the shear tab.

Shear tab plate

Option	Description
Shear plate	Plate thickness.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Shear tab dimensions



	Description
1	Horizontal size of the cut created for the shear tab. Equal size is created on both sides of the shear tab.

	Description
2	Vertical size of the cut created for the shear tab. Equal size is created at the top and bottom of the shear tab.
3	Distance from the center line of the shear tab to the center line of the main part. Default value `0` places the shear tab at the center line of the main part.
4	Corner radius of the cut created for the shear tab.
5	Vertical dimension of the shear tab chamfer.

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Concave arc chamfer

Shear tab cut

When the component is created near the column top, the upper edge of the shear tab can be extended to the top of the column.

Option	Description
	Default Line AutoDefaults can change this option.
	Line
	Column top The upper edge of the shear tab is extended to the top of the main part.

Shear tab shape

Option	Description
	Default Perpendicular to the main part AutoDefaults can change this option.
	Perpendicular to the main part
	Automatic
	Shear tab edges are aligned in the direction of the right secondary part.
	Shear tab edges are aligned in the direction of the left secondary part.
	The edges of the shear tab are connected at the intersection of the secondary part edges.
	Defined by both edges

Shear tab orientation

Define the shear tab orientation for both secondary parts.

Option	Option	Description
		Default Sloped AutoDefaults can change this option.
		Automatic Sloped or square If the secondary part is sloped less than 10 degrees, a square shear tab is created. Otherwise, the shear tab is sloped in the direction of the secondary part.
		Sloped
		Square

Option	Option	Description
		Sloped with square end cut

Shear tab top corner shape

Define the shape of the shear tab top corner for both secondary parts.

Option	Option	Description
		Default Bevel AutoDefaults can change this option.
		Bevel
		Square

Shear tab bottom corner shape

Define the shape of the shear tab bottom corner for both secondary parts.

Option	Option	Description
		Default Bevel AutoDefaults can change this option.
		Bevel
		Square

Shear tab position

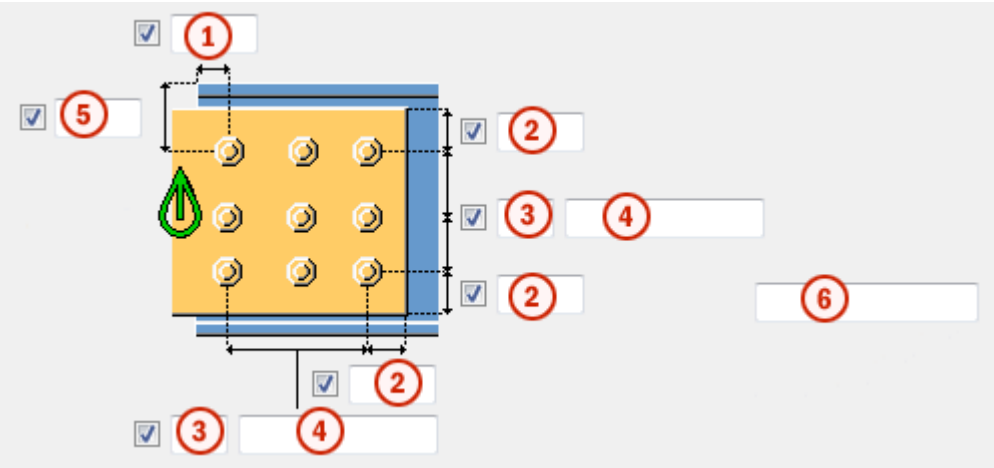
Option	Description
	Default Near side AutoDefaults can change this option.
	Automatic The shear tab is created to the near side when the angle between the main part and the secondary part is less than 90 degrees.

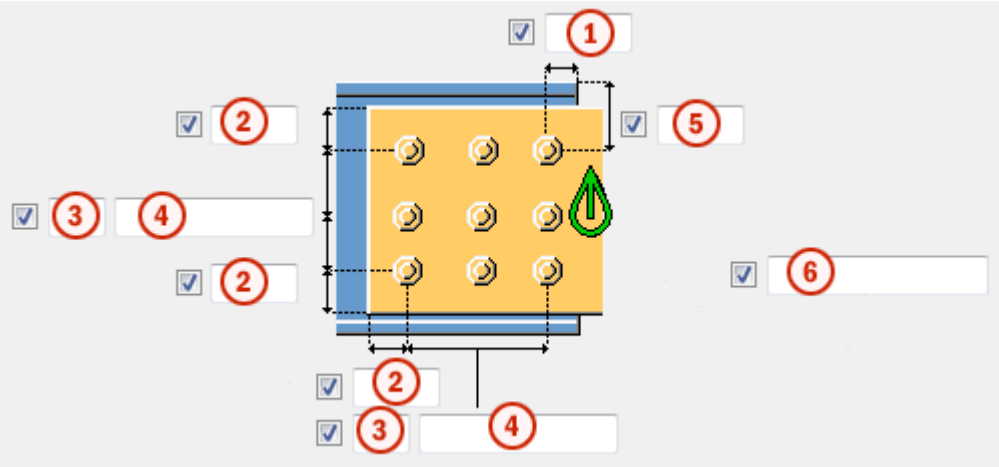
Option	Description
	Near side
	Both sides
	Far side

1stSecBolts and 2ndSecBolts tabs

Use the **1stSecBolts** tab and the **2ndSecBolts** tab to control the properties of the bolts that connect the shear tab to the first and the second secondary part.

Bolt group dimensions





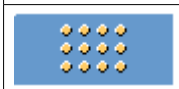
	Description
1	Dimension for horizontal bolt group position.

	Description
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position.
6	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Bolt group orientation

Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Staggered Bolts are staggered in the direction of the secondary part.
	Square Square bolt group is positioned horizontally.
	Sloped Square bolt group is sloped in the direction of the secondary part.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered

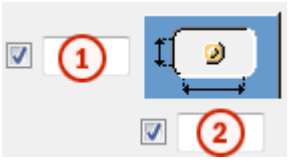
Option	Description
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



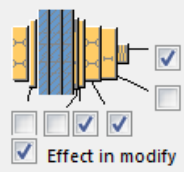
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.

Option	Description	Default
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

5.2 Clip angle connections

This section introduces clip angle connection components available in Tekla Structures.

Click the links below to find out more:

- Angle cleat (3) (page 1164)
- Two sided angle cleat (25) (page 1168)
- Clip angle (116) (page 1172)
- Two sided clip angle (117) (page 1182)
- Clip angle (141) (page 1194)
- Two sided clip angle (143) (page 1240)

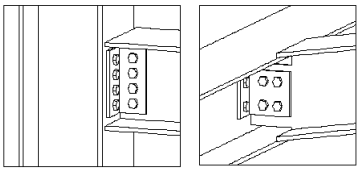
Angle cleat (3)

Angle cleat (3) connects two beams or a beam to a column using one or two bolted clip angles.

Objects created

- Clip angle (L profile)
- Bolts
- Welds

Use for

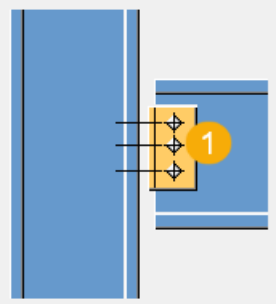
Situation	Description
	Clip angle bolted between the parts. The secondary part can be sloped.

Selection order

1. Select the main part.
2. Select the secondary part.

   The connection is created automatically when the secondary part is selected.

Part identification key

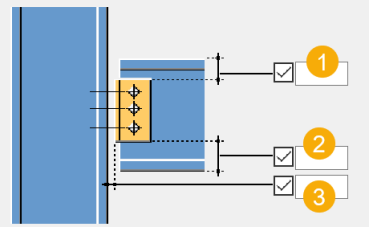


	Description
1	Clip angle

Picture tab

Use the **Picture** tab to define the clip angle length, and the gap between the main part and the clip angle.

Dimensions



	Description
1	Clip angle edge distance from the secondary part upper edge. You can control the clip angle length by defining the distance from the secondary part upper and lower edge.
2	Clip angle edge distance from the secondary part lower edge.
3	Gap between the main part and the clip angle. By default, the value is `0`.

Parts tab

Use the **Parts** tab to define the clip angle properties.

Parts

Option	Description	Default
L profile	Select the profile from the profile catalog.	`L150-100-10`

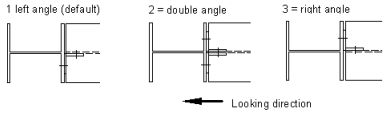
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

Use the **Parameters** tab to define the location of the clip angles.

Placing

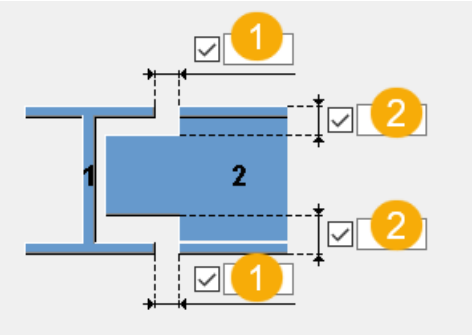
Option	Description
Placing	By default, the clip angle is created on the left side. **Middle** creates two clip angles.



Notch tab

Use the **Notch** tab to define the secondary part notch dimensions.

Cut dimensions



	Description	
1	Dimension for the horizontal flange cut.	`10 mm`
2	Dimension for the vertical flange cut.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest `5 mm`.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Bolts

Click the link below to find out more:

Welds

Click the link below to find out more:

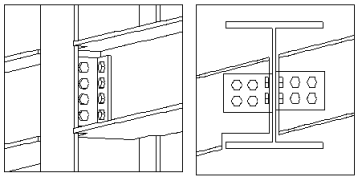
Two sided angle cleat (25)

Two sided angle cleat (25) connects two beams to a beam or to a column with one or two bolted clip angles.

Objects created

- Clip angles
- Bolts

Use for

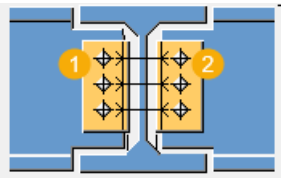
Option	Description
	Two beams connected to a column or a beam with bolted clip angles.

Selection order

1. Select the main part (column or beam).
2. Select the first secondary part (beam).
3. Select the second secondary part (beam).

   Click the middle mouse button to create the component.

Part identification key

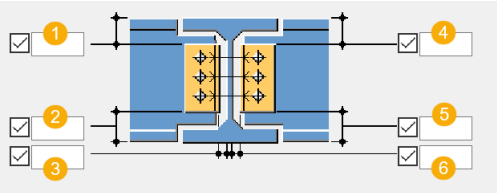


	Description
1	L profile
2	L profile 2

Picture tab

Use the **Picture** tab to define the L profile length and the gap between the main part and the angle profiles. You can control the L profile length with the L profile edge distance.

Dimensions



	Description
1	L profile edge distance from the first secondary part upper edge.
2	L profile edge distance from the first secondary part lower edge.
3	Gap between the main part and the angle profile in the first secondary part. By default, the value is zero.
4	L profile 2 edge distance from the second secondary part upper edge.
5	L profile 2 edge distance from the second secondary part lower edge.
6	Gap between the main part and the angle profile in the second secondary part. By default, the value is zero.

Parts tab

Use the **Parts** tab to define the angle profile properties.

Parts

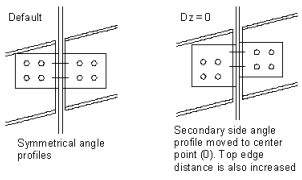
Option	Description	Default
L profile	Select the profile from the profile catalog.	L150-100-10
L profile 2	Select the profile from the profile catalog.	L150-100-10

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

Use the **Parameters** tab to define the position of the angle profiles.

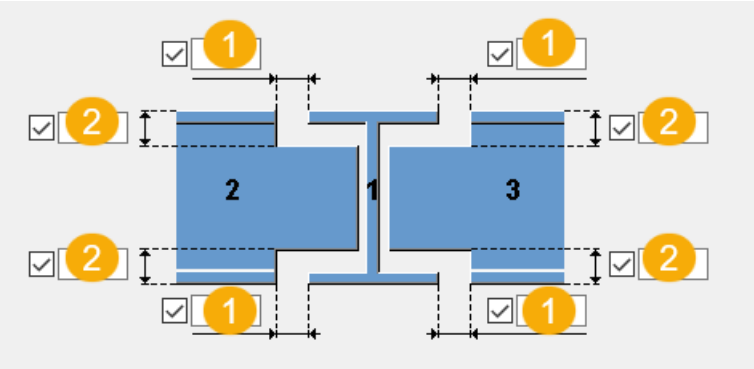
Angle profile positions

Option	Description
Placing, **Placing 2**	Define the number and side of angles in single angle connections separately for both sides.
Move L-prof. 2 in z direction, **Move L-prof. 2 in y direction** 	Move the position of angle profiles on the secondary part side. By default, the angle profiles are symmetrical. For example, when connecting skewed secondary beams, the offsets can be used for placing the angle profiles.

Notch tab

Use the **Notch** tab to define the secondary part notch dimensions.

Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	`10 mm`
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest `5 mm`.

Bolts

Click the link below to find out more:

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Design tab

Click the link below to find out more:

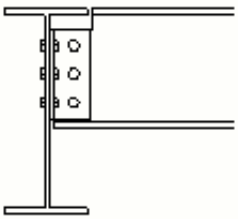
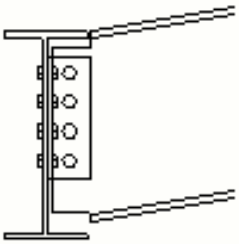
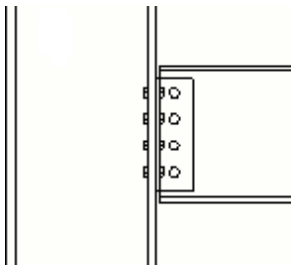
Clip angle (116)

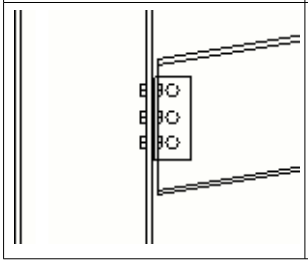
Clip angle (116) connects a beam to a beam or a column with a clip angle.

Objects created

- Clip angle (1 or 2)
- Bolts
- Cuts

Use for

Situation	Description
	Clip angle connection to a beam web.
	Clip angle connection to a beam web. The secondary beam is sloped.
	Clip angle connection to a column flange.

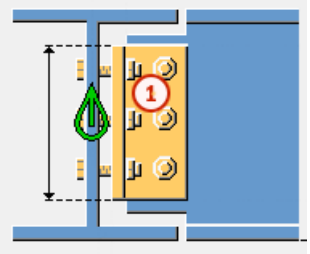
Situation	Description
	Clip angle connection to a column flange. The secondary beam is sloped.

Selection order

1. Select the main part (beam or column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

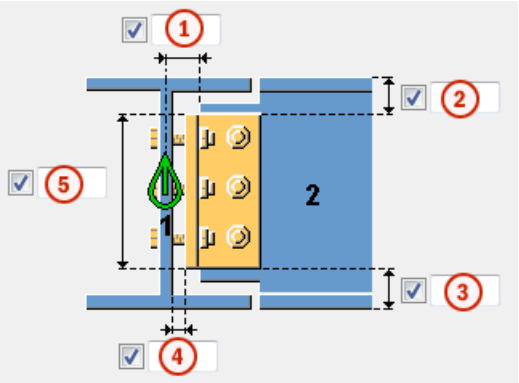


	Part
1	Clip angle

Picture tab

Use the **Picture** tab to control the clip angle dimensions and the clip angle position.

Dimensions



	Description	Default
1	Cut length for the secondary part.	
2	Clip angle upper edge distance from the top of the secondary beam. The upper edge position of the angle modifies the clip angle height. Positive value moves the top position closer to the beam center and thus decreases the clip angle size. Negative values increase the clip angle size.	If no value is entered, bolts and bolt edge distances define the size of the clip angle.
3	Clip angle lower edge distance from the bottom of the secondary beam. The lower edge position of the angle modifies the clip angle height. Positive value moves the top position closer to the beam center and thus decreases the clip angle size. Negative values increase the clip angle size.	If no value is entered, bolts and bolt edge distances define the size of the clip angle.
4	Gap between the main part and the clip angle.	
5	Height of the clip angle.	

Clip angle position

Option	Description
	Default Near and far side clip angles are created. AutoDefaults can change this option.
	Near side clip angle is created.
	Near side and far side clip angles are created.
	Far side clip angle is created.

Parts tab

Use the **Parts** tab to control the properties of the clip angle.

Clip angle

Part	Description
L profile	Define the clip angle profile by selecting it from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Notch tab

Use the **Notch** tab to create notches for the secondary beam and to control the notch properties.

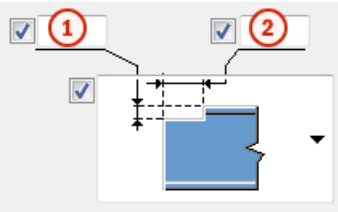
BCSA notch definition

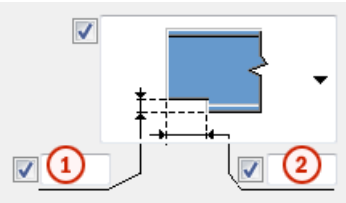
Define whether the notch is created according to British Constructional Steelwork Association (BCSA) specifications.

Option	Description
Default	Notch dimensions.
Yes	Creates a 50 mm notch for simple beam-to-beam connections.
No	Use the options on this **Notch** tab to define the notch dimensions.

Notch dimensions

Define the top and the bottom dimensions of the notch if you have set the **BCSA notch def** option to **No**.

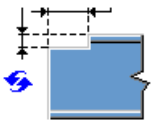
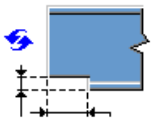
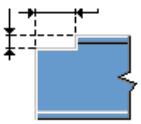
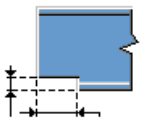


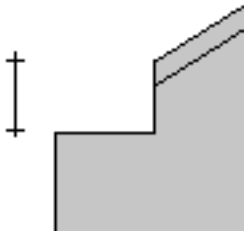
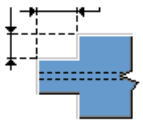
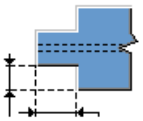
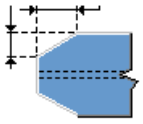
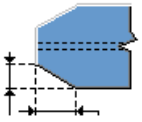
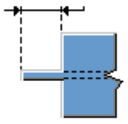
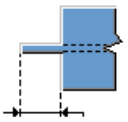
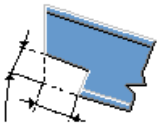


	Description
1	Vertical notch dimension.
2	Horizontal notch dimension.

Notch shape

Define the notch shape for the top and the bottom of the secondary beam.

Option	Option	Description
		Default Creates a square notch on the top side or on the bottom side of the secondary beam. AutoDefaults can change this option.
		No notch
		Creates a square notch on the top side or on the bottom side of the secondary beam. Define the notch dimensions. In beam-to-beam connections with a sloped secondary beam,

Option	Option	Description
		the depth is measured as shown in the picture. 
		Creates a notch on both sides of the secondary part. Define the notch dimensions.
		Creates a chamfered notch on both sides of the secondary beam. Define the chamfer dimensions.
		Creates a strip. Define the length of the strip. The flanges are cut completely.
		Creates a special type of square notch. Define the notch dimensions. The notch is square to the secondary beam. There are no default values for the length or the depth.

Notching side

Define on which side of the secondary beam the notch is created. You can define the side for both the top and the bottom of the secondary beam.

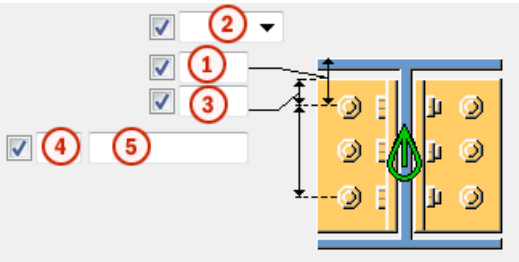
Option	Description
	Default Creates notches on both sides. AutoDefaults can change this option.

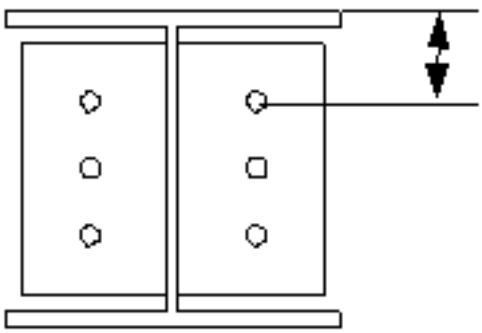
Option	Description
	Creates notches on both sides.
	Creates a notch on the left side.
	Creates a notch on the right side.

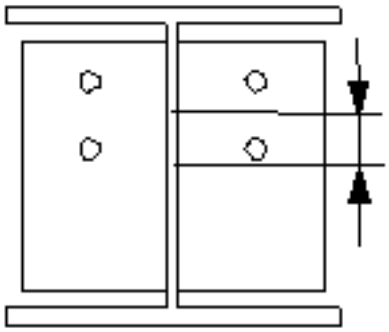
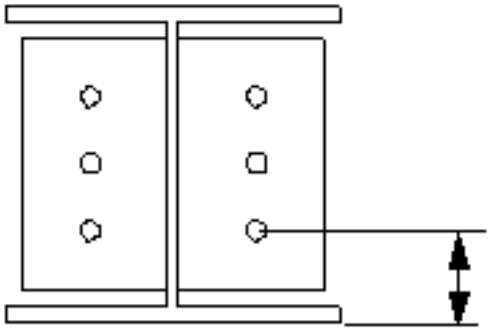
Bolts tab

Use the **Bolts** tab to control the bolt properties. The vertical position of bolts in the main part must be aligned with the vertical position of bolts in the secondary part. Vertical bolts cannot be staggered.

Bolt group dimensions



	Description
1	Dimension for vertical bolt group position.
2	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

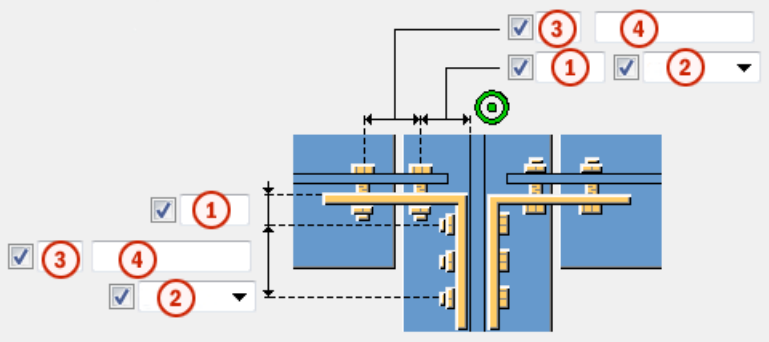
	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered

Option	Description
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt group dimensions



	Description
1	Bolt edge distance.
2	Location where the bolts should be attached.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	

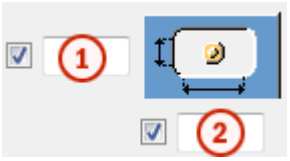
Option	Description	Default
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Slotted holes

You can define slotted, oversized, or tapped holes.

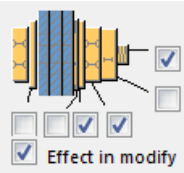


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

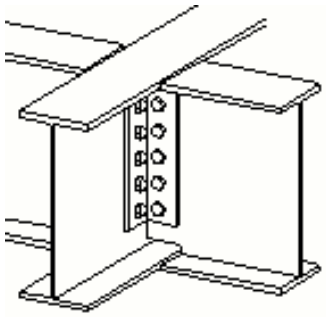
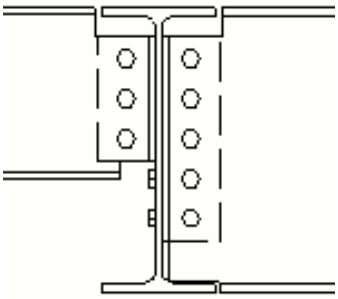
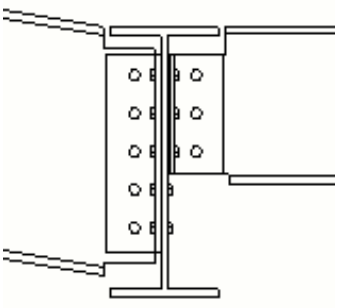
Two sided clip angle (117)

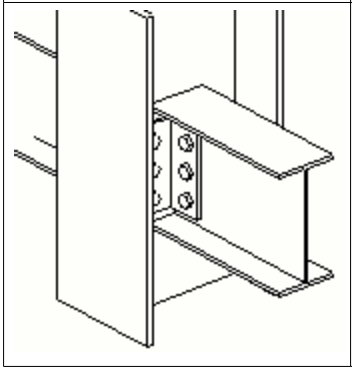
Two sided clip angle (117) connects two beams to a beam or a column using clip angles. The clip angles are bolted to the secondary beams and to the main part.

Objects created

- Clip angles (2 or 4)
- Bolts
- Cuts

Use for

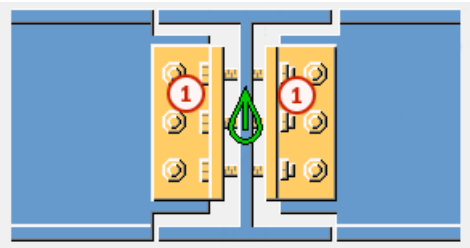
Situation	Description
	Clip angle connection to a beam web.
	Clip angle connection to a beam web. Two secondary beams with different heights.
	Clip angle connection to a beam web. The other secondary beam is sloped.

Situation	Description
	Clip angle connection to a column web.

Selection order

1. Select the main part (beam or column).
2. Select the first secondary part (beam).
3. Select the second secondary part (beam).
4. Click the middle mouse button to create the connection.

Part identification key

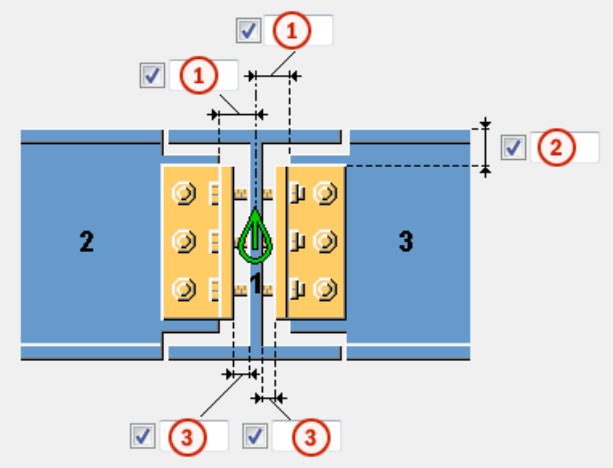


	Part
1	Clip angle

Picture tab

Use the **Picture** tab to control the dimensions and the position of the clip angle.

Dimensions



	Description	Default
1	Cut length for the secondary part.	`2.25 mm`
2	Clip angle upper edge distance from the top of the secondary beam. The upper edge position of the angle modifies the clip angle height. Positive value moves the top position closer to the beam center and thus decreases the clip angle size. Negative values increase the clip angle size.	If no value is entered, bolts and bolt edge distances define the size of the clip angle.
3	Gap between the main part and the clip angle.	

Clip angle position

Option	Description
	Default Near side and far side clip angles are created. AutoDefaults can change this option.
	Near side clip angles are created.
	Near side and far side clip angles are created.
	Far side clip angles are created.

Parts tab

Use the **Parts** tab to control the clip angle properties.

Clip angle

Part	Description
L profile, **L profile 2**	Define the clip angle profile by selecting it from the profile catalog.
Angle 1 length, **Angle 2 length**	Define the length of the clip angle both on the first secondary part and the second secondary part side.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Move cleats to smaller web

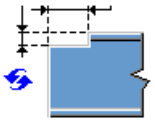
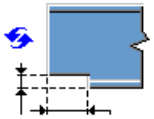
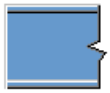
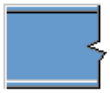
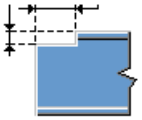
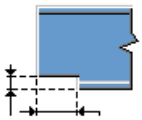
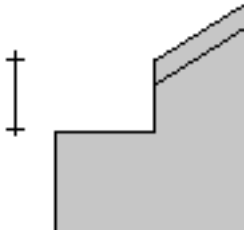
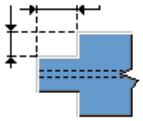
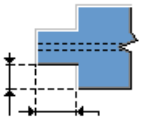
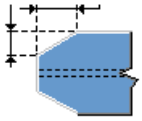
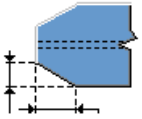
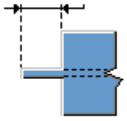
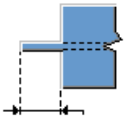
Define the position of the cleats.

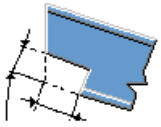
Notch tab

Use the **Notch** tab to create notches for the secondary beams and to control the notch properties. Define the notches for both secondary beams.

Notch shape

Define the notch shape for the top and the bottom of the secondary beam.

Option	Option	Description
		Default Creates a square notch on the top side or on the bottom side of the secondary beam. AutoDefaults can change this option.
		No notch
		Creates a square notch on the top side or on the bottom side of the secondary beam. Define the notch dimensions. In beam-to-beam connections with a sloped secondary beam, the depth is measured as shown in the picture. 
		Creates a notch on both sides of the secondary part. Define the notch dimensions.
		Creates a chamfered notch on both sides of the secondary beam. Define the chamfer dimensions.
		Creates a strip. Define the length of the strip. The flanges are cut completely.

Option	Option	Description
		Creates a special type of square notch. Define the notch dimensions. The notch is square to the secondary beam. There are no default values for the length or the depth.

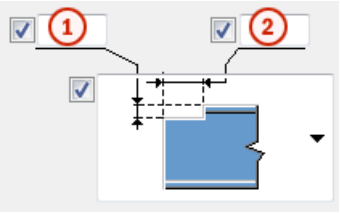
Notching side

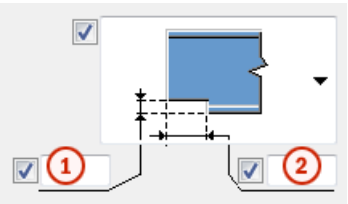
Define on which side of the secondary beam the notch is created. You can define the side for both the top and the bottom of the secondary beam.

Option	Description
	Default Creates notches on both sides. AutoDefaults can change this option.
	Creates notches on both sides.
	Creates a notch on the left side.
	Creates a notch on the right side.

Notch dimensions

Define the top and the bottom dimensions of the notch if you have set the **BCSA notch def** option to **No**.





	Description
1	Vertical notch dimension.
2	Horizontal notch dimension.

BCSA notch definition

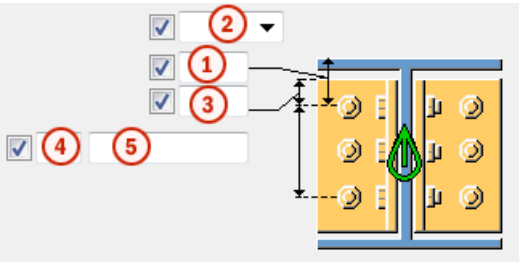
Define whether the notch is created according to British Constructional Steelwork Association (BCSA) specifications.

Option	Description
Default	Notch dimensions.
Yes	Creates a 50 mm notch for simple beam-to-beam connections.
No	Use the options on this **Notch** tab to define the notch dimensions.

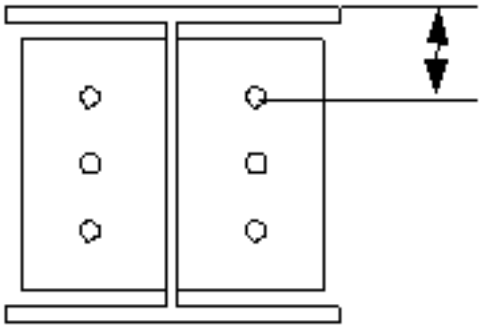
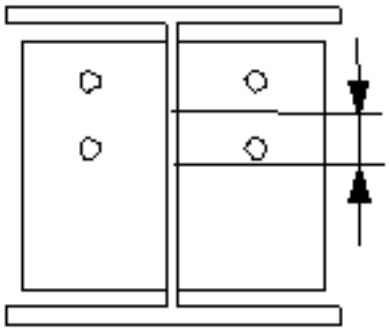
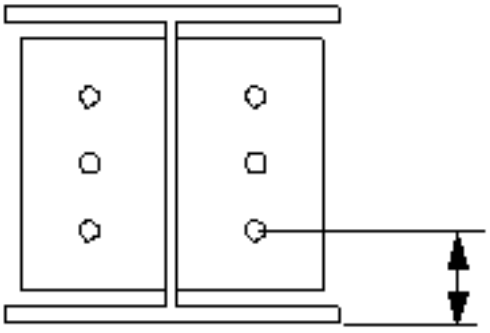
Bolts tab

Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions



	Description
1	Dimension for vertical bolt group position.

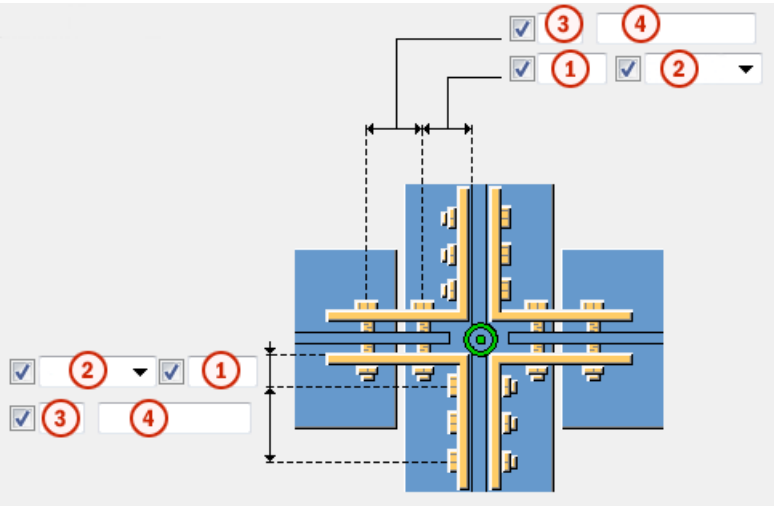
	Description
2	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

	Description
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt group dimensions



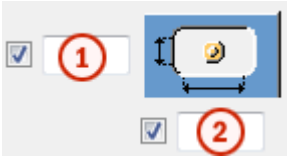
	Description
1	Bolt edge distance.
2	Location where the bolts should be attached.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



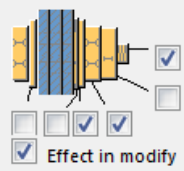
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

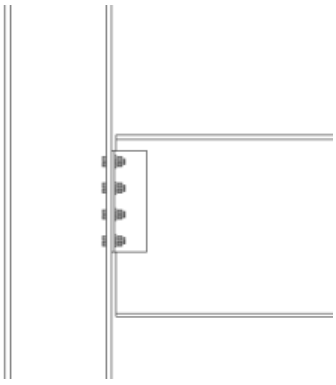
Clip angle (141)

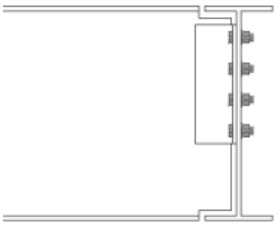
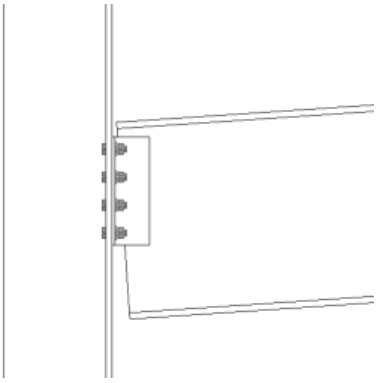
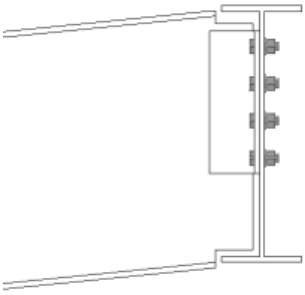
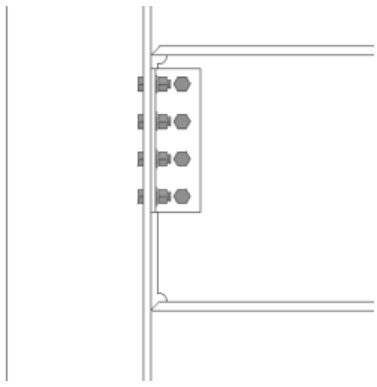
Clip angle (141) connects two beams, or a beam to a column, using bolted or welded clip angles. The secondary beam can be leveled or sloped. Welded haunch plates and seat angles are optional.

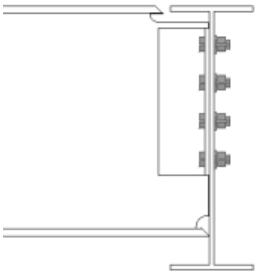
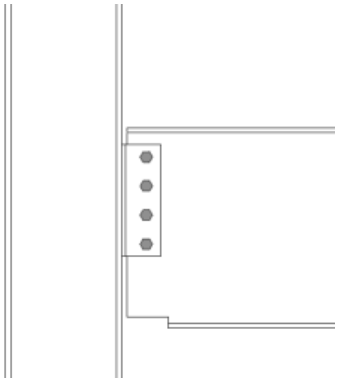
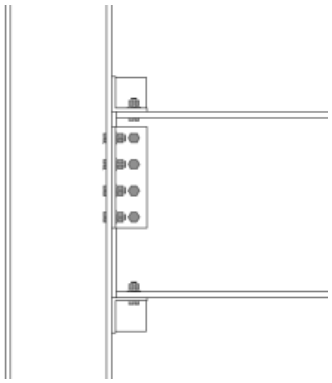
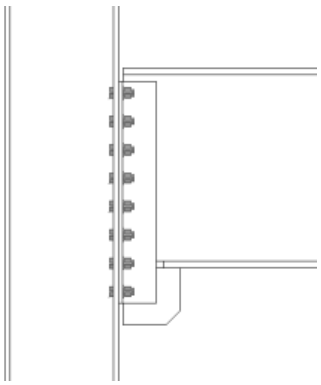
Objects created

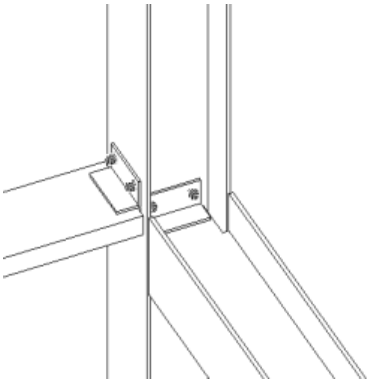
- Clip angles (1 or 2)
- Stiffeners (optional)
- Top and bottom haunch plate (optional)
- Seat angle (optional)
- Seat stiffeners (optional)
- Bolts
- Plate washers (optional)
- Welds
- Cuts

Use for

Situation	Description
	Clip angle connection to column flange or web. Single-sided/double-sided clip. Welded/bolted, bolted/bolted, welded/welded options.

Situation	Description
	Clip angle connection to beam web. Single-sided/double-sided clip.
	Clip angle connection to column flange or web. Single-sided/double-sided clip. The secondary part is sloped. The secondary part end can be cut square or bevel.
	Clip angle connection to beam web. Single-sided/double-sided clip. The secondary part is sloped. There are several notching options.
	Clip angle connection to column flange or web. Single-sided/double-sided clip. Weld preparation and weld access holes for moment connection.

Situation	Description
	Clip angle connection to beam web. Single-sided/double sided clip. Option for weld preparation.
	Clip angle connection to column flange. Bottom flange is blocked or stripped for erection.
	Clip angle connection to column. Seat angle option. Top/Bottom/Both.
	Clip angle connection to column flange or web. Single-sided/double-sided clip. Haunch option. Top/Bottom/Both.

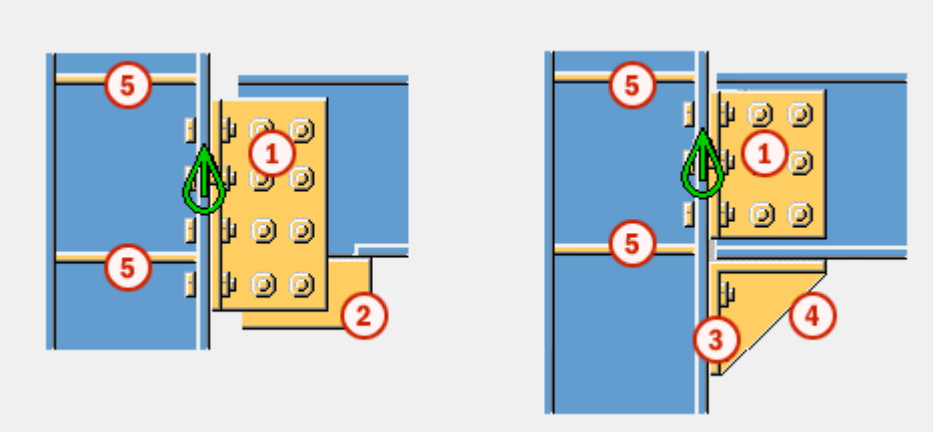
Situation	Description
	Clip angle connection to column flange or web. Single-sided/double-sided clip. Secondary part is rotated.

Selection order

1. Select the main part (beam or column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key



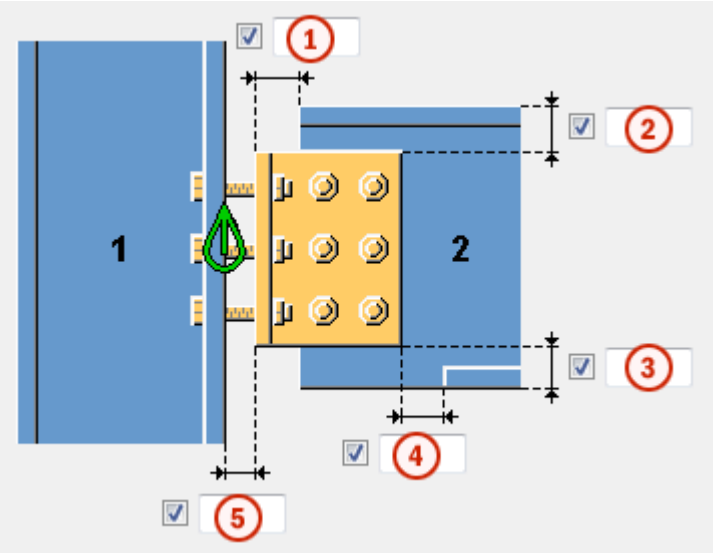
	Part
1	Clip angles
2	Haunch plate
3	Seat angle
4	Seat stiffener
5	Web stiffeners

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to control the clip angle dimensions and to define how the beam end is cut.

Dimensions



	Description	Default
1	Cut length for the secondary part. The cut is defined from the clip angle edge.	20 mm
2	Clip angle upper edge distance from the top of the secondary beam. The upper edge position of the angle modifies the clip angle height. Positive value moves the top position closer to the beam center and thus decreases the clip angle size. Negative values increase the clip angle size.	If no value is entered, bolts and bolt edge distances define the size of the clip angle.
3	Clip angle lower edge distance from the bottom of the secondary beam. The lower edge position of the angle modifies the clip angle height. Positive value moves the top position closer to the beam center and thus decreases the clip angle size. Negative values increase the clip angle size.	If no value is entered, bolts and bolt edge distances define the size of the clip angle.

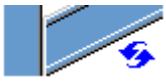
	Description	Default
4	Size of the strip made to the secondary part flange. The cutting point of the flange is defined from the clip angle edge.	The flange is automatically stripped when the clip angle crosses the flange. 10 mm
5	Define the gap between the main part and the clip angle.	0

Beam bottom flange cut

Option	Description
	Default Flange cut AutoDefaults can change this option.
	Notch The bottom of the secondary beam is notched if the clip angle crosses the flange. Enter the notch radius and height.
	Flange cut The secondary beam flange is cut on the same side as the clip angle if the clip angle crosses the flange.

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.
	Square Cuts the end of the secondary beam square.

Option	Description
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.

Parts tab

Use the **Parts** tab to control the size, position and orientation of the clip angles.

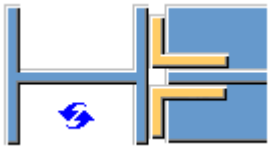
Profile NS/FS

Option	Description	Default
Profile NS	Near side clip angle profile by selecting it from the profile catalog.	Angle size is defined by the bolt diameter. The default name is `ANGLE`.
Profile FS	Far side clip angle profile by selecting it from the profile catalog.	Angle size determined by the bolt diameter. The default name is `ANGLE`.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

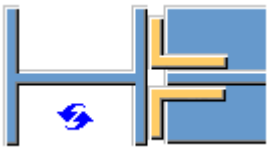
Clip angle position

Define the number of clip angles and the side of the clip angle in single clip angle connections.

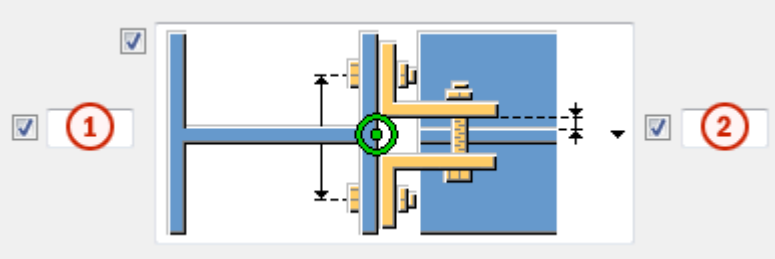
Option	Description
	Default Near side and far side clip angles are created. AutoDefaults can change this option.
	Automatic If the main part is a tube profile, two clip angles are created. Otherwise a near side clip angle is created.
	Near side clip angle is created.
	Near side and far side clip angles are created.
	Far side clip angle is created.
	Wrapped angle. Far side clip angle is created.
	Wrapped angle. Near side clip angle is created.

Clip angle orientation

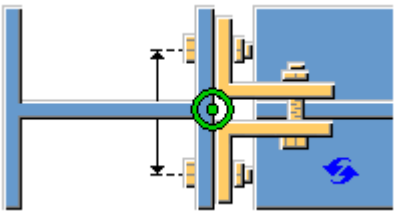
These options switch the unequal legs of the far side and the near side clip angles.

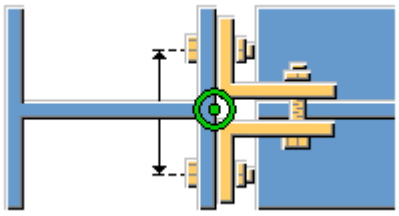
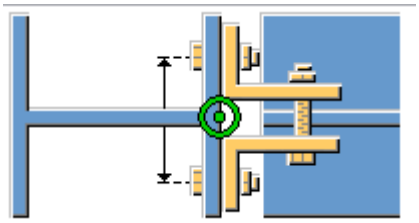
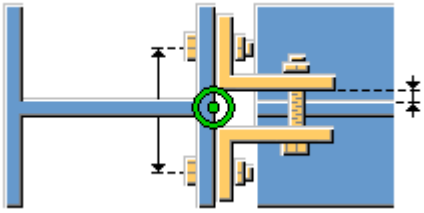
Option for near side	Option for far side	Description
		Default Not switched AutoDefaults can change this option.
		Not switched The clip angle is placed on a connection so that the longer leg is connected to the secondary part.
		Switched The clip angle legs are switched so that the longer leg is connected to the main part.

Bolt spacing and weld gap

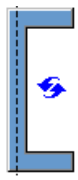
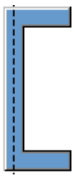
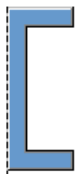


Option	Description
1	Bolt spacing.
2	Weld gap.

Option	Description
	Default Bolt spacing without weld gap AutoDefaults can change this option.

Option	Description
	Bolt spacing without weld gap
	Bolt spacing with weld gap
	Bolt spacing with weld gap that you can define

Center of the bolt gage line

Option	Description
	Default Center of the bolt gage line from the center line of the secondary part. AutoDefaults can change this option.
	Center of the bolt gage line from the center line of the secondary part.
	Center of the bolt gage line from the back of the secondary part web. You can use this option when the secondary part has a C or U profile.

Stiffeners tab

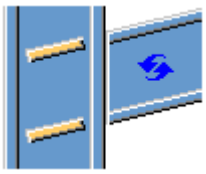
Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position, and type.

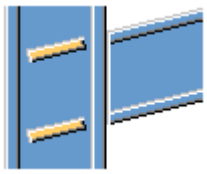
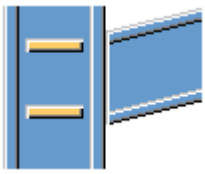
Stiffener plate dimensions

Option	Description
Top NS	Top near side stiffener thickness, width and height.
Top FS	Top far side stiffener thickness, width and height.
Bottom NS	Bottom near side stiffener thickness, width and height.
Bottom FS	Bottom far side stiffener thickness, width and height.

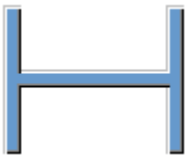
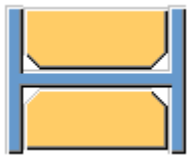
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Stiffener orientation

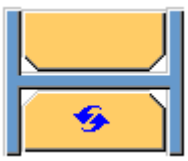
Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.

Option	Description
	Stiffeners are parallel to the secondary part.
	Stiffeners are perpendicular to the main part.

Stiffener creation

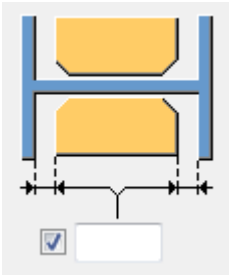
Option	Description
	Default Stiffeners are created. AutoDefaults can change this option.
	Automatic Stiffeners are created when necessary.
	No stiffeners are created.
	Stiffeners are created.

Stiffener shape

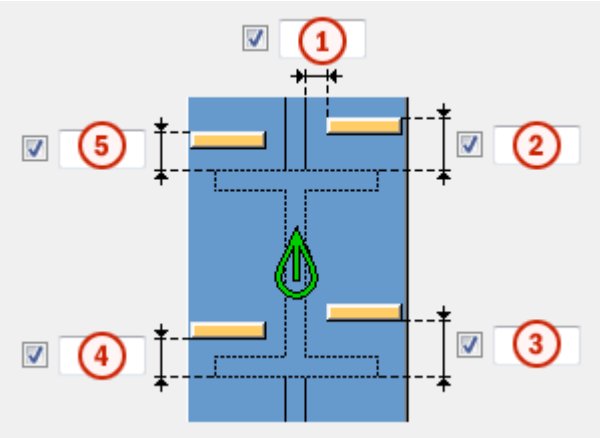
Option	Description
	Default Line chamfered stiffener plates AutoDefaults can change this option.

Option	Description
	Automatic Line chamfered stiffener plates
	Square stiffener plates Stiffener plates with a gap for the main part web rounding
	Line chamfered stiffener plates

Stiffener gap

Option	Description
	Size of the gap between the flanges and the stiffener.

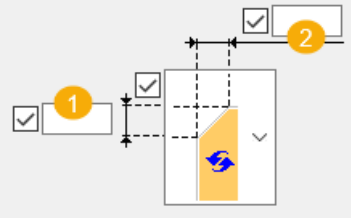
Stiffener positions



	Description
1	Size of the gap between the stiffener and the beam web edge.

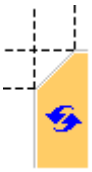
	Description
2	Size of the gap between the top near side stiffener and the beam flange edge.
3	Size of the gap between the bottom near side stiffener and the beam flange edge.
4	Size of the gap between the bottom far side stiffener and the beam flange edge.
5	Size of the gap between the top far side stiffener and the beam flange edge.

Chamfer dimensions



	Description	Default
1	Vertical dimension of the chamfer.	`10 mm`
2	Horizontal dimension of the chamfer.	`10 mm`

Chamfer type

Option	Description
	Default. Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer

Option	Description
	Convex arc chamfer
	Concave arc chamfer

Haunch tab

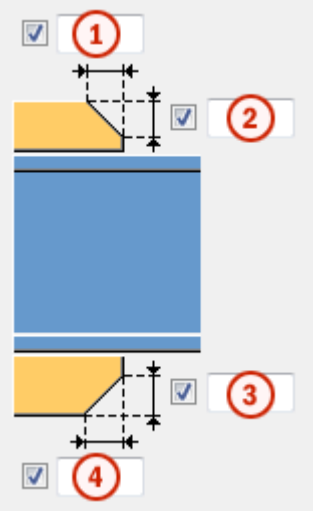
Use the **Haunch** tab to control the haunch plate creation and chamfers in the secondary beam flanges.

Haunch plates

Option	Description
Top plate	Top haunch plate thickness, width and height.
Bottom plate	Bottom haunch plate thickness, width and height.

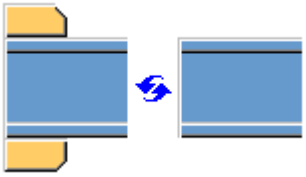
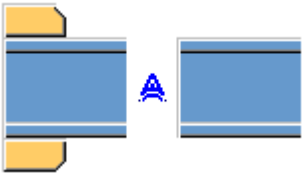
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Haunch plate chamfers



	Description
1	Width of the top haunch plate chamfer.
2	Height of the top haunch plate chamfer.
3	Height of the bottom haunch plate chamfer.
4	Width of the bottom haunch plate chamfer.

Haunch plate creation

Option	Description
	Default Top and bottom haunch plates are created, if needed. AutoDefaults can change this option.
	Automatic Top or bottom haunch plate or both are created, if needed.
	Top and bottom haunch plates are created. To create a single plate, enter `0` in the thickness (**t**) field for the plate you do not need (top or bottom plate).

Option	Description
	Haunch plates are not created.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beams and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

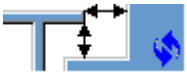
Automatic notching options affect both the top and the bottom flange.

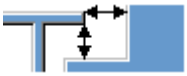
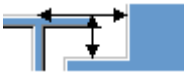
Notch shape

Automatic notching is switched on when you select a notch shape.

Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

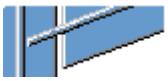
Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.

Option	Description
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

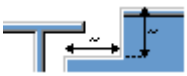
Enter the horizontal and vertical values for the cuts.

Flange cut shape

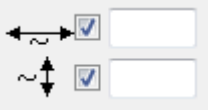
Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.



Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.

Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

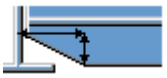
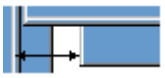
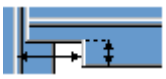
Side of flange notch

The side of flange notch defines on which side of the beam the notches are created.

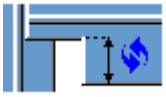
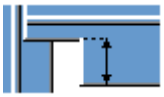
Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

Flange notch shape

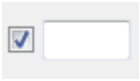
The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

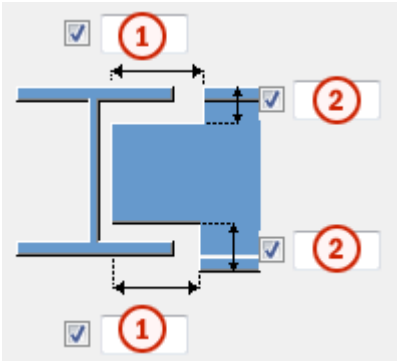
Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.



Cut dimensions

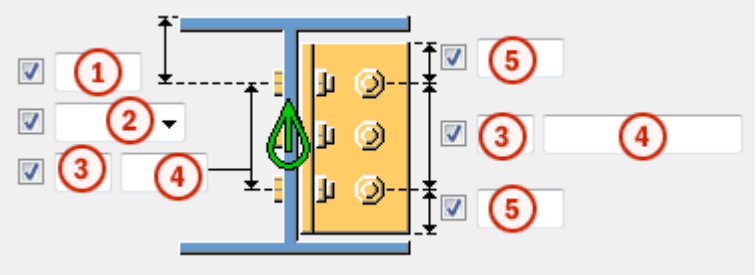


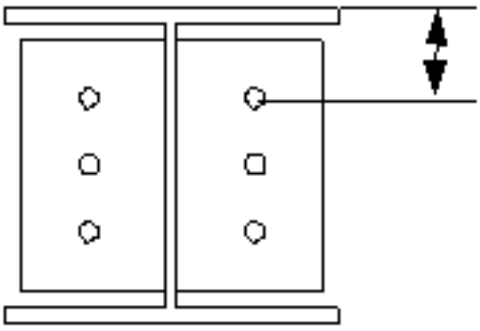
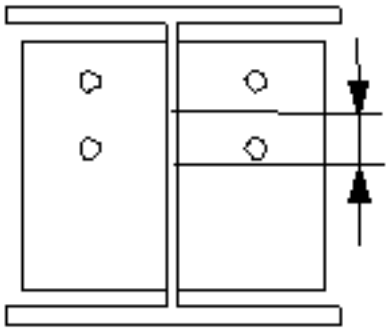
	Description	Default
1	Dimensions for the horizontal flange cuts.	`10 mm`
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest `5 mm`.

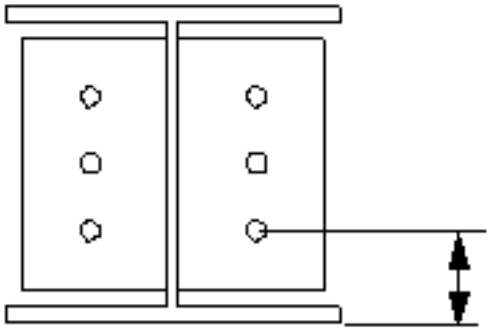
Bolts tab

Use the **Bolts** tab to control bolts and welds that connect the clip angle to the main part and to the secondary part.

Bolt group dimensions



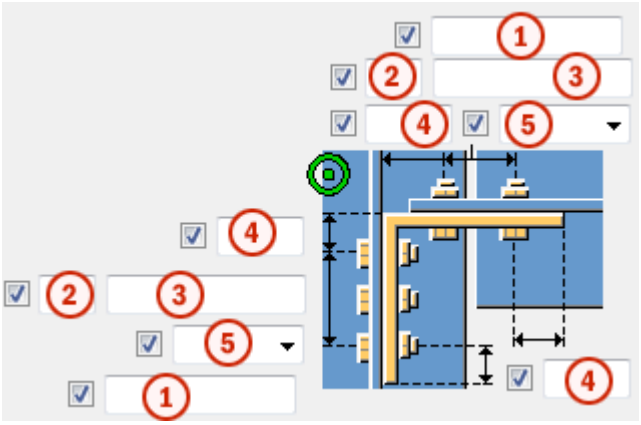
	Description
1	Dimension for vertical bolt group position.
2	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part. 

	Description
	• **Below**: From the lower edge of the secondary part to the lowest bolt. 
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.

Staggering of bolts on clip angle

Option	Description
	Default Bolts are not staggered. AutoDefaults can change this option.
	Bolts are not staggered. The bolts that connect the clip angle to the secondary part are on the same horizontal level as the bolts that connect the clip angle to the main part.
	Bolts on the main part are staggered. The bolts that connect the clip angle to the main part are moved downwards by half the bolt vertical spacing value.
	Bolts on the secondary part are staggered. The bolts that connect the clip angle to the secondary part are moved downwards by half the bolt vertical spacing value.
	Bolts on the secondary part are staggered. The bolts that connect the clip angle to the sloped secondary part are parallel to the secondary part.

Bolt group dimensions



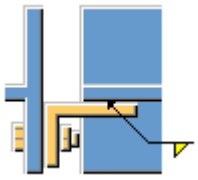
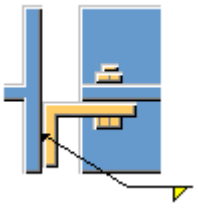
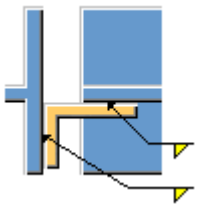
	Description
1	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Bolt edge distance.
5	Location where the bolts should be attached.

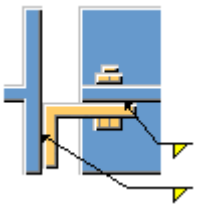
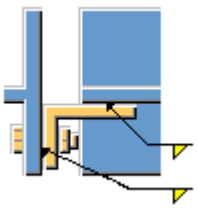
Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3

Option	Description
	Staggered type 4

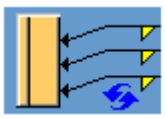
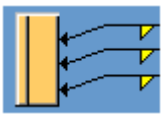
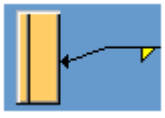
Attachment type

Option	Description
	Default Both parts are bolted. AutoDefaults can change this option.
	Automatic When the main part is a tube profile, the clip angles are welded to the main part and bolted to the secondary part. Otherwise the clip angles are bolted to both parts.
	Both parts are bolted.
	Main part is welded and secondary is part bolted.
	Main part is bolted and secondary part is welded.
	Both parts are welded.

Option	Description
	Main part is not bolted.
	Secondary part is not welded.
	Secondary part is not bolted.
	Both parts are bolted and welded.

Number of clip angle welds

Define the number of welds that connect the clip angle to the main part and/or the secondary part.

Option	Description
	Default Three welds are created to the clip angle. AutoDefaults can change this option.
	Three welds are created to the clip angle.
	One weld is created to the clip angle.

Bolt basic properties

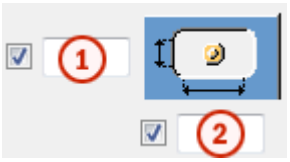
Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Slotted holes

You can define slotted, oversized, or tapped holes.



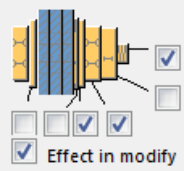
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Plate washers tab

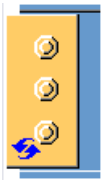
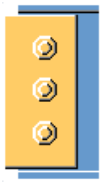
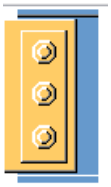
Use the **Plate washers** tab to define the plate washer properties to the main part and to the secondary parts.

Option	Description
Plate washer	Plate washer thickness, width and height.

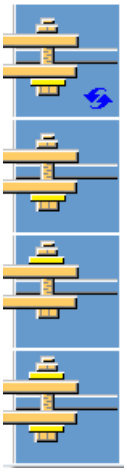
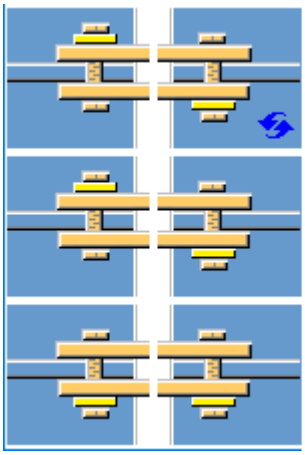
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Plate washer

Define plate washers for bolts and select the plate washer side.

Option	Description
	Default No plate washer AutoDefaults can change this option.
	No plate washer
	One plate washer

Option	Description
	Individual square plate washers for each bolt
	Individual round plate washers for each bolt

Option	Description
	Select whether the plate washer is created for one clip angle or both clip angles.
	Select whether the plate washers are placed symmetrically or asymmetrically.

Beam cut tab

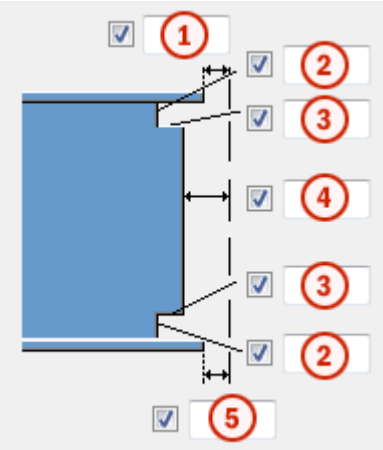
Use the **Beam cut** tab to control weld backing bars, weld access holes, beam end preparations, and flange cuts.

Weld backing bar

Option	Description
Weld backing bar	Weld backing bar thickness and width.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

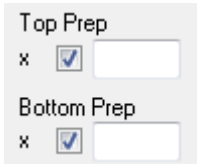
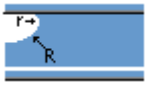
Weld access hole dimensions



	Description
1	Gap between the secondary part top flange and the main part.
2	Vertical dimensions for the top and the bottom weld access holes.

	Description
3	Horizontal dimensions for the top and the bottom weld access holes.
4	Gap between the secondary part web and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.
5	Gap between the secondary part bottom flange and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.

Weld access holes

Option	Description	Default
	Default Round weld access hole AutoDefaults can change this option.	
	Round weld access hole	
	Square weld access hole	
	Diagonal weld access hole	
	Round weld access hole with a radius that you can define in 	
	Extended cone-shaped weld access hole with a radius and dimensions that you can define in and 	
	Cone-shaped weld access hole with radiuses that you can	`R = 35` `r = 10`

Option	Description	Default
	define in R ☑ and r ☑ Capital **R** defines the large radius (height). Small **r** defines the small radius.	

Beam end preparation

Option	Description
	Default Top and bottom flange are prepared. AutoDefaults can change this option.
	Automatic Top and bottom flange are prepared.
	Beam end is not prepared.
	Top and bottom flange are prepared.
	Top flange is prepared.
	Bottom flange is prepared.

Flange cut

Option for top flange	Option for bottom flange	Description
		Default Flange is not cut. AutoDefaults can change this option.
		Flange is not cut.

Option for top flange	Option for bottom flange	Description
		Flange is cut.

Weld backing bars

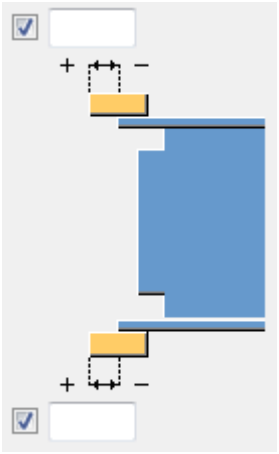
Option for top backing bar	Option for bottom backing bar	Description
		Default Backing bars are created inside the flanges. AutoDefaults can change this option.
		No backing bars are created.
		Backing bars are created inside the flanges.
		Backing bars are created outside the flanges.

Weld backing bar length

Enter the length of the weld backing bar in the box below the options.

Option	Description
	Default Absolute length of the backing bar AutoDefaults can change this option.
	Absolute length of the backing bar
	Extension beyond the edge of the flange

Weld backing bar position

Option	Description
	Enter a positive or a negative value to move the front end of the backing bar relative to the end of the flange.

Assembly type

Define the location where the weld backing bar welds are made. When you select the **Workshop** option, Tekla Structures includes the backing bars in the assembly.

Angle box tab

Use the **Angle box** tab to add a seat angle.

Seat angle

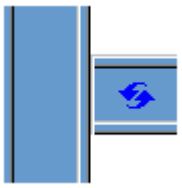
The purpose of seat angles is to carry loads from the secondary part. Seat angles can be positioned to top, bottom or both flanges of the secondary part. The seat angle can be stiffened, and bolted or welded to the main and secondary parts.

Option	Description
Stiffeners	Stiffener thickness, width and height.
Top angle, **Bottom angle**	Selea the seat angle profile by selecting it from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

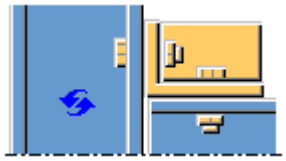
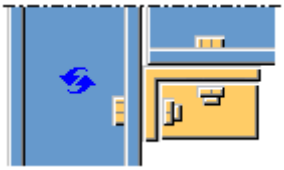
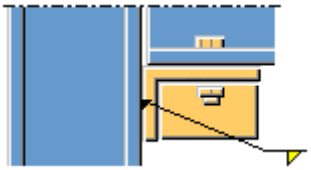
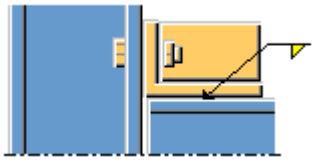
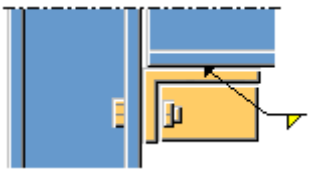
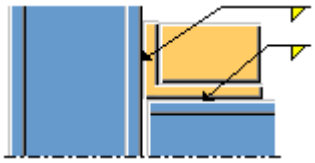
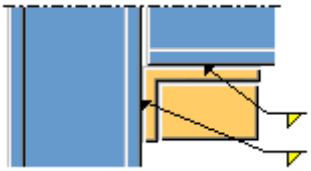
Option	Description	Default
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Seat angle position

Option	Description
	Default No seat angle is created. AutoDefaults can change this option.
	No seat angle is created.
	Seat angle is created at the top of the flange.
	Seat angle is created at the bottom of the flange.
	Seat angles are created on both sides of the flange.

Seat angle attachment

Seat angle is positioned at the top or at the bottom of the secondary part.

Option for top seat angle	Option for bottom seat angle	Description
		Default Bolted Seat angle is bolted to the main part and to the secondary part. AutoDefaults can change this option.
		Bolted Seat angle is bolted to the main part and to the secondary part.
		Welded-bolted Seat angle is welded to the main part and bolted to the secondary part.
		Bolted-welded Seat angle is bolted to the main part and welded to the secondary part.
		Welded Seat angle is welded to the main part and to the secondary part.

Stiffener type

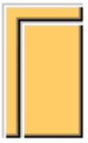
Option	Description
	Default Rectangular stiffener plate AutoDefaults can change this option.
	Rectangular stiffener plate

Option	Description
	Triangular stiffener plate
	The line connecting the ends of the seat angle legs defines the stiffener plate shape.

Seat angle rotation

Option	Description
	Default Seat angle is not rotated. AutoDefaults can change this option.
	Seat angle is not rotated.
	Seat angle is rotated horizontally 90 degrees. To stiffen the rotated seat angle, select the **Middle stiffeners** option in the **Middle stiffener position** list.

Seat angle orientation

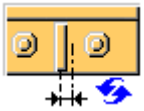
Option	Description
	Default The longer leg of the seat angle is connected to the secondary part. AutoDefaults can change this option.
	The longer leg of the seat angle is connected to the secondary part.
	The longer leg of the seat angle is connected to the main part.
	Automatic The longer leg of the seat angle is connected to the part where bolts

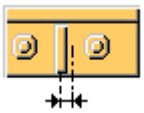
Option	Description
	reach furthest from the seat angle corner.

Side stiffener position

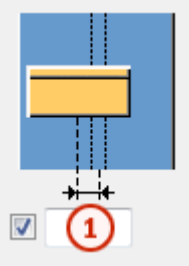
Option	Description
	Default No side stiffeners are created. AutoDefaults can change this option.
	No side stiffeners are created.
	Near side stiffeners are created.
	Far side stiffeners are created.
	Near side and far side stiffeners are created.

Middle stiffener position

Option	Description
	Default According to bolts AutoDefaults can change this option.
	No middle stiffener plate is created.
	Middle stiffeners The stiffener plate is positioned in the middle of the seat angle. Enter the number of middle stiffeners in the **Number of middle stiffeners** box. Multiple stiffeners are centered and equally spaced.

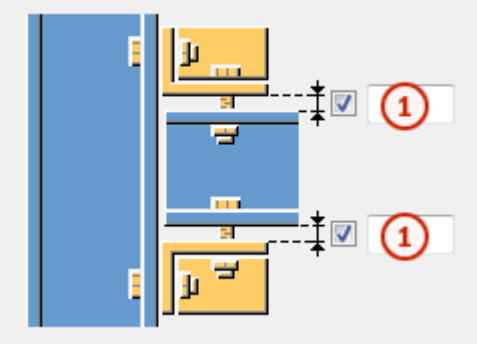
Option	Description
	According to bolts The stiffener plate is positioned between the bolts in the middle of the bolt spacing. By default, stiffener is created between every two bolts. Enter the number of middle stiffeners in the box below the **According to bolts** option.

Seat angle offset



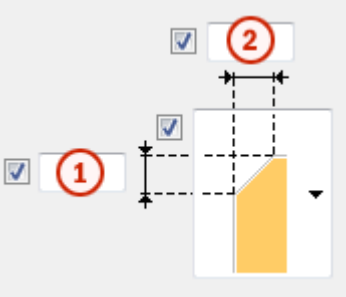
	Description
1	Seat angle horizontal offset from the center line of the main part.

Gap



	Description
1	Top gap and bottom gap between the seat angle and the secondary part.

Chamfer dimensions



	Description
1	Vertical dimension of the chamfer.
2	Horizontal dimension of the chamfer.

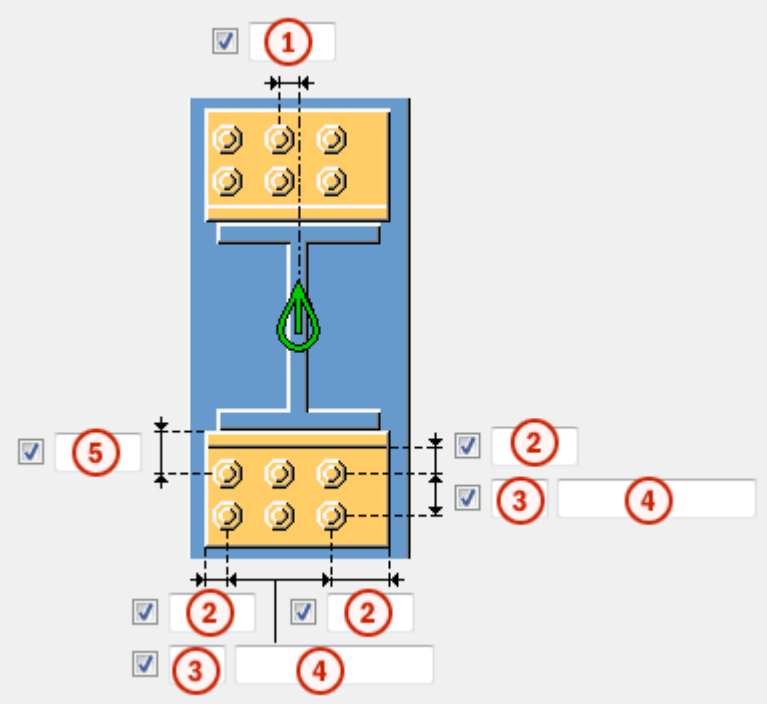
Chamfer type

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

BoxPBolts tab

Use the **BoxPBolts** tab to control properties of the bolts that connect the seat angle to the main part.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position. The dimension is defined from the middle line of the secondary beam.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position. The dimension is defined from the bottom of the secondary beam.

Top

Top refers to the bolt group that connects the top seat angle to the main part.

Bottom

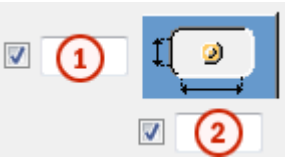
Bottom refers to the bolt group that connects the bottom seat angle to the main part.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

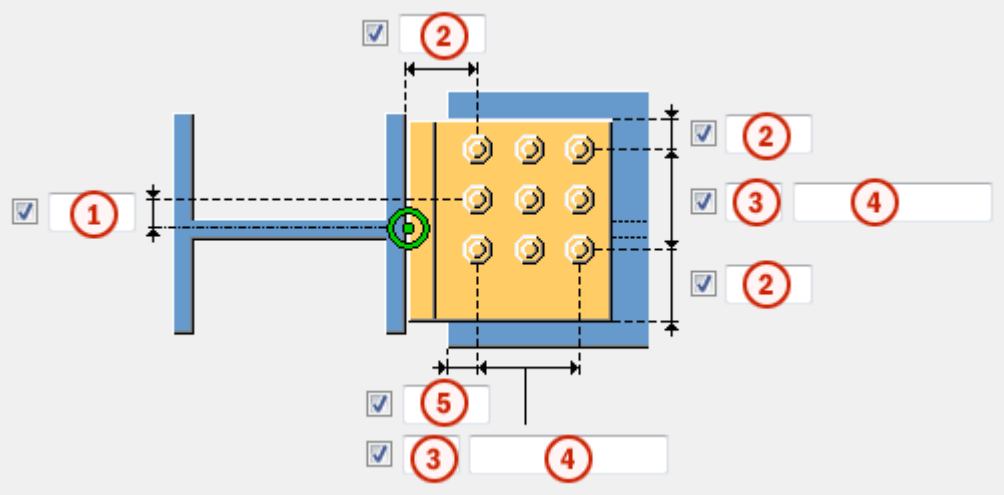
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

BoxSBolts tab

Use the **BoxSBolts** tab to control properties of the bolts that connect the seat angle to the secondary part.

Bolt group dimensions



	Description
1	Dimension for vertical bolt group position. The dimension is defined from the middle line of the secondary beam.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for horizontal bolt group position. The dimension is defined from the bottom of the secondary beam.

Top

Top refers to the bolt group that connects the top seat angle to the secondary part.

Bottom

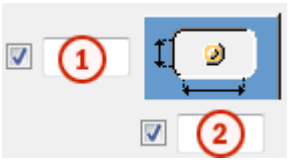
Bottom refers to the bolt group that connects the bottom seat angle to the secondary part.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Design type tab

Click the link below to find out more:

Design type tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

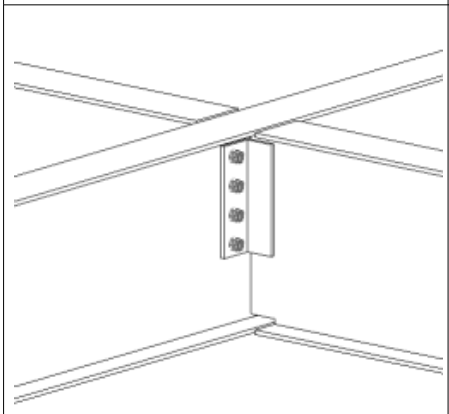
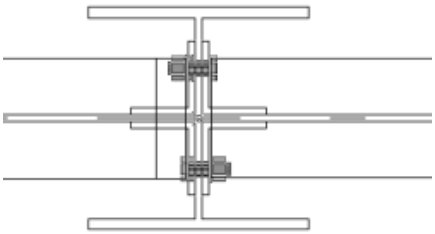
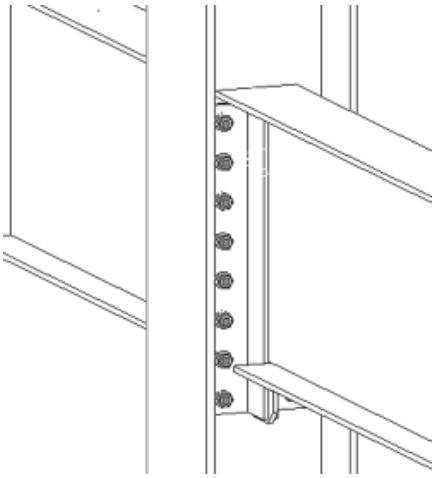
Two sided clip angle (143)

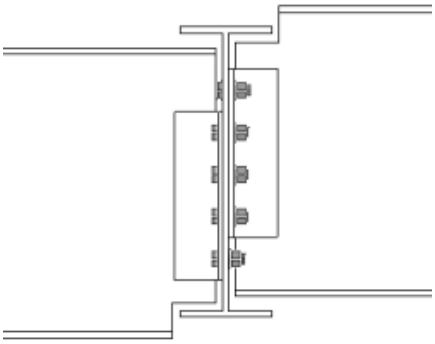
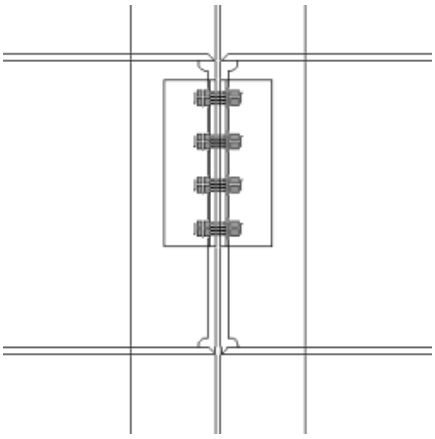
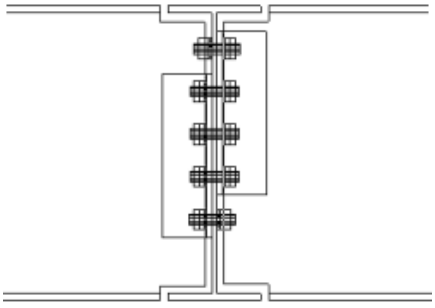
Two sided clip angle (143) connects two beams, or a beam to a column, using single or double clip angles. The secondary beam can be leveled or sloped. Welded haunch plates and a seat angle are optional.

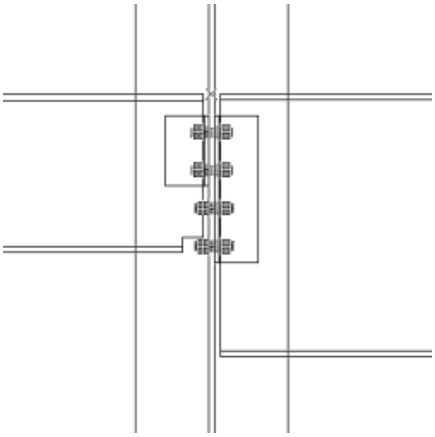
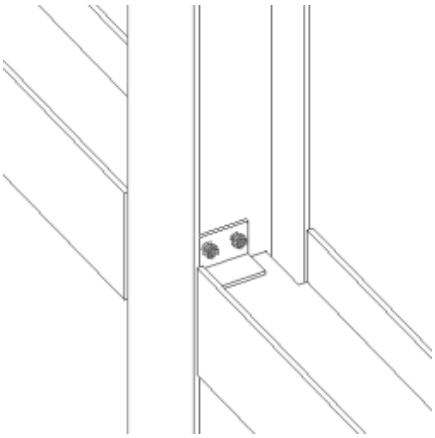
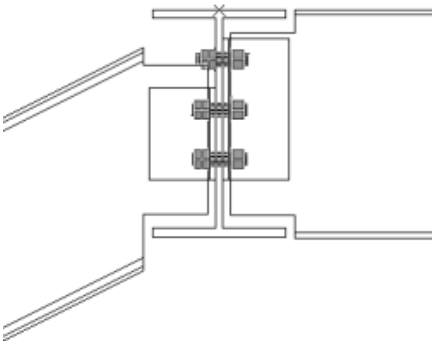
Objects created

- Clip angles (2)
- Haunch plates (optional)
- Seat angle (optional)
- Weld backing bars (optional)
- Bolts
- Plate washers (optional)
- Welds
- Cuts

Use for

Situation	Description
	Clip angle connection. Single-sided/double-sided clip. Two secondary parts. Bolted/bolted, welded/bolted, welded/welded options.
	Clip angle connection. Single-sided/double-sided clip. Two secondary parts. Bolted/bolted, welded/bolted, welded/welded options.
	Clip angle connection. Single-sided/double-sided clip. Two secondary parts. Haunch option. Top/Bottom/Both

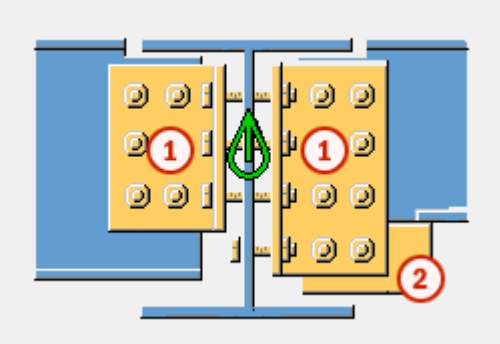
Situation	Description
	Clip angle connection. Single-sided/double-sided clip. Two secondary parts at different heights.
	Clip angle connection. Single-sided/double-sided clip. Two secondary parts. Weld preparation and weld access holes for moment connection.
	Clip angle connection. Single-sided/double-sided clip. Two secondary parts. Safety connection.

Situation	Description
	Clip angle connection. Automatic notching of secondary part to provide bolt clearance.
	Clip angle connection. Single-sided/double-sided clip. Two secondary parts rotated.
	Clip angle connection. Single-sided/double-sided clip. Two secondary parts, one sloped.

Selection order

1. Select the main part (column or beam).
2. Select the first secondary part (beam).
3. Select the second secondary part (beam).
4. Click the middle mouse button to create the connection.

Part identification key



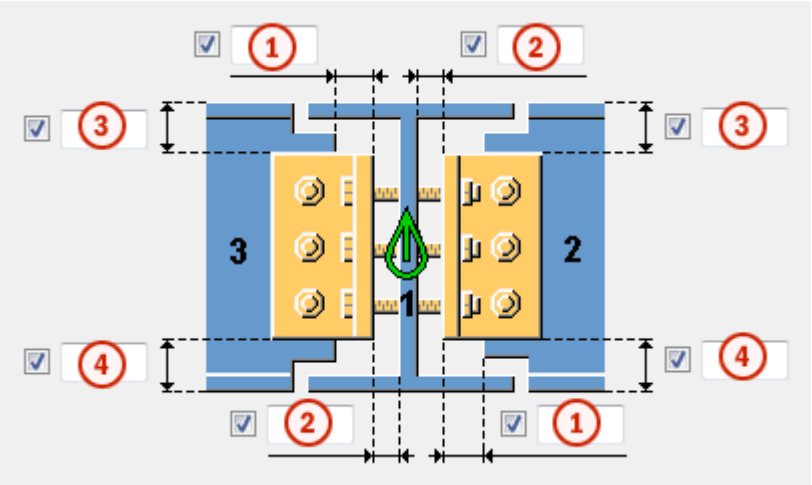
	Part
1	Clip angles
2	Haunch plate

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to control the clip angle dimensions and to define how the beam end is cut.

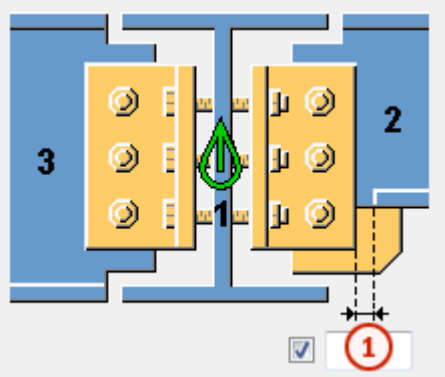
Dimensions



	Description	Default
1	Cut length for the secondary part. The cut is defined from the clip angle edge.	
2	Define the gap between the main part and the clip angle.	0

	Description	Default
3	Clip angle upper edge distance from the top of the secondary beam. The upper edge position of the angle modifies the clip angle height. Positive value moves the top position closer to the beam center and thus decreases the clip angle size. Negative values increase the clip angle size.	If no value is entered, bolts and bolt edge distances define the size of the clip angle.
4	Clip angle lower edge distance from the bottom of the secondary beam. The lower edge position of the angle modifies the clip angle height. Positive value moves the top position closer to the beam center and thus decreases the clip angle size. Negative values increase the clip angle size.	If no value is entered, bolts and bolt edge distances define the size of the clip angle.

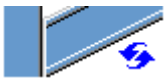
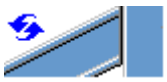
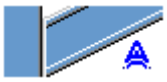
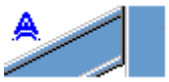
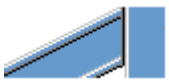
Option	Description
Sort secondaries by profile height	Select whether the secondary parts are sorted by their profile height. When set to **Yes**, the highest secondary part is always regarded as the first secondary part, regardless of the selection order.



	Description	Default
1	Size of the strip made to the secondary part flange. The cutting point of the flange is defined from the clip angle edge.	The flange is automatically stripped when the clip angle crosses the flange.

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Option	Description
		Default Bevel AutoDefaults can change this option.
		Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.
		Square Cuts the end of the secondary beam square.
		Bevel Cuts the end of the secondary beam parallel to the edge of the main part.

Beam bottom flange cut

Option	Description
	Default Flange cut AutoDefaults can change this option.
	Notch The bottom of the secondary beam is notched if the clip angle crosses the flange. Enter the notch radius and height.
	Flange cut The secondary beam flange is cut on the same side as the clip angle if the clip angle crosses the flange.

Parts tab

Use the **Parts** tab to control the size, position and orientation of the clip angles.

Profile NS/FS

Option	Description	Default
Profile NS **Profile 2 NS**	Near side clip angle profile for the first and the second secondary parts by selecting it from the profile catalog.	Angle size is defined by the bolt diameter. The default name is `ANGLE`.
Profile FS **Profile 2 FS**	Far side clip angle profile for the first and the second secondary parts by selecting it from the profile catalog.	Angle size is determined by the bolt diameter. The default name is `ANGLE`.

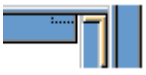
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Clip angle position

Option for the second secondary part	Option for the first secondary part	Description
		Default Near side and far side clip angles are created. AutoDefaults can change this option.
		Automatic If the main part is a tube profile, two clip angles are created. Otherwise a near side clip angle is created.
		Near side clip angle is created.
		Near side and far side clip angles are created.
		Far side clip angle is created.

Near side clip angle orientation

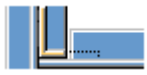
These options switch the unequal legs of the near side clip angles.

Option for the second secondary part	Option for the first secondary part	Description
		Default Not switched AutoDefaults can change this option.
		Not switched The clip angle is placed on a connection so that the longer leg is connected to the secondary part.
		Switched The clip angle legs are switched so that the

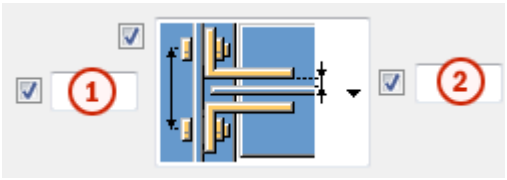
Option for the second secondary part	Option for the first secondary part	Description
		longer leg is connected to the main part.

Far side clip angle orientation

These options switch the unequal legs of the far side clip angles.

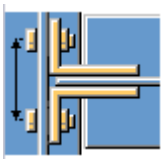
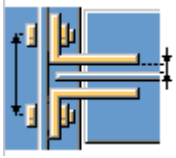
Option for the second secondary part	Option for the first secondary part	Description
		Default Not switched AutoDefaults can change this option.
		Not switched The clip angle is placed on a connection so that the longer leg is connected to the secondary part.
		Switched The clip angle legs are switched so that the longer leg is connected to the main part.

Bolt spacing and weld gap

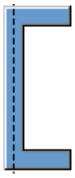
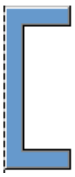


Option	Description
1	Bolt spacing.
2	Weld gap.

Option	Description
	Default Bolt spacing without weld gap AutoDefaults can change this option.

Option	Description
	Bolt spacing without weld gap
	Bolt spacing with weld gap
	Bolt spacing with weld gap that you can define.

Center of the bolt gage line

Option	Description
	Default Center of the bolt gage line from the center line of the secondary part. AutoDefaults can change this option.
	Center of the bolt gage line from the center line of the secondary part.
	Center of the bolt gage line from the back of the secondary part web. You can use this option when the secondary part has a C or U profile.

Safety connections

In **Side 2 horizontal offset**, enter a value to the field to move the clip angles in horizontal direction on the side of the second secondary beam. Enter the **Vertical cut/offset** and **Horizontal cut** dimensions.

Safety connection type

Define whether the created connection is a standard connection or a safety connection.

Option	Description
	Default No safety connection is created. AutoDefaults can change this option.
	No safety connection is created.
	Clip angles are moved down to create safety connection. Enter the dimension in the **Vertical cut/offset** field.
	Clip angles are notched to create safety connection. Enter the notch dimensions in the **Vertical cut/offset** and **Horizontal cut** fields.

Safety connection location

Define the location of the safety connection.

Option	Description
	Default On the first secondary beam AutoDefaults can change this option.
	On the first secondary beam
	On the first secondary beam
	On the second secondary beam
	On the second secondary beam
	On the first secondary beam

Option	Description
	On the second secondary beam
	On the first secondary beam and the second secondary
	On the second secondary beam and the first secondary beam

Safety connection staggering

Define how the clip angles are staggered.

Option	Description
	Default Staggered AutoDefaults can change this option.
	Staggered Clip angle height is not modified, but the clip angles are moved to create safety connection.
	Flush Clip angle height is modified to create safety connection, but the bottoms of the angles are flush with each other.

Haunch tab

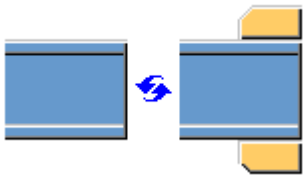
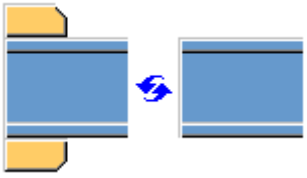
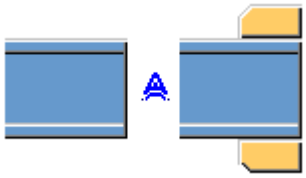
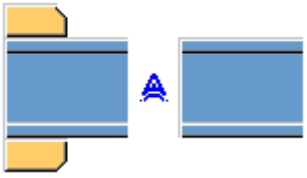
Use the **Haunch** tab to control the haunch plate creation and chamfers in the secondary beam flanges.

Haunch plates

Option	Description
Top plate	Top haunch plate thickness, width and height.
Bottom plate	Bottom haunch plate thickness, width and height.

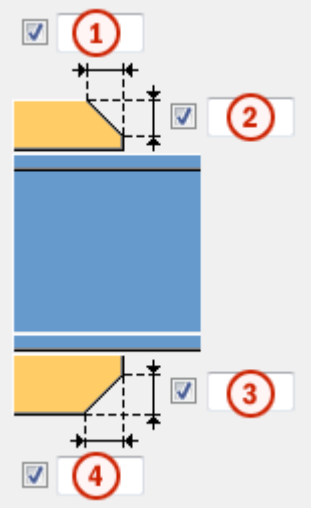
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Haunch plate creation

Option for the second secondary beam	Option for the first secondary beam	Description
		Default Top and bottom haunch plates are created, if needed. AutoDefaults can change this option.
		Automatic Top or bottom haunch plate or both are created, if needed.
		Top and bottom haunch plates are always created. To create a single plate, enter 0 in the thickness (**t**) field for the haunch plate you do not need (top or bottom plate).

Option for the second secondary beam	Option for the first secondary beam	Description
		Haunch plates are not created.

Haunch plate chamfers



	Description
1	Width of the top haunch plate chamfer.
2	Height of the top haunch plate chamfer.
3	Height of the bottom haunch plate chamfer.
4	Width of the bottom haunch plate chamfer.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beams and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

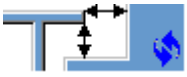
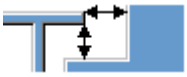
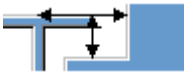
Automatic notching options affect both the top and the bottom flange.

Notch shape

Automatic notching is switched on when you select a notch shape.

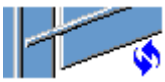
Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

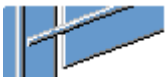
Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

Enter the horizontal and vertical values for the cuts.

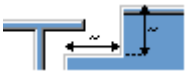
Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.

Option	Description
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.

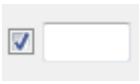
Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.



Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

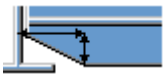
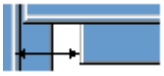
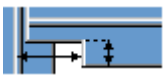
Side of flange notch

The side of flange notch defines on which side of the beam the notches are created.

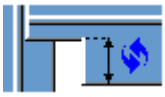
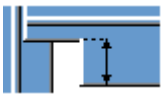
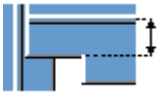
Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

Flange notch shape

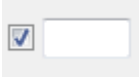
The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

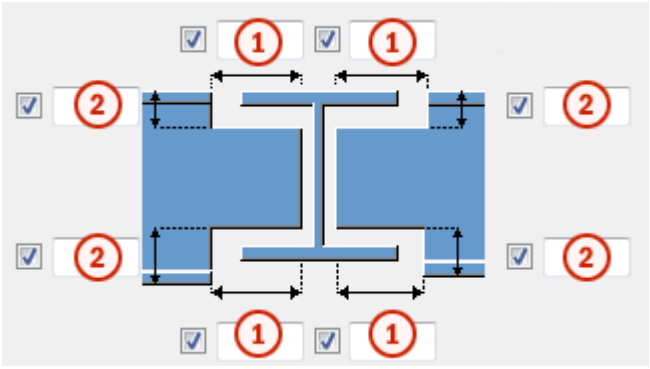
Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.



Cut dimensions

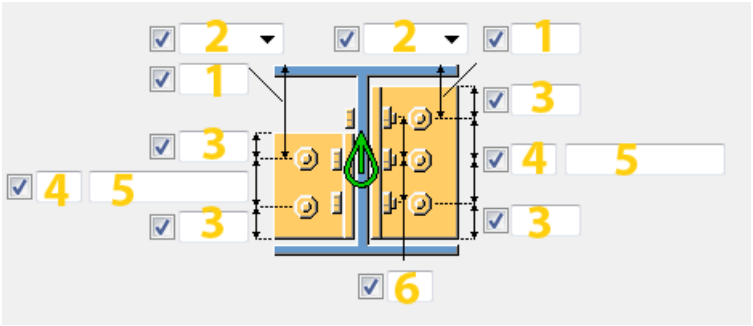


	Description	Default
1	Dimensions for the horizontal flange cuts.	`10 mm`
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest `5 mm`.

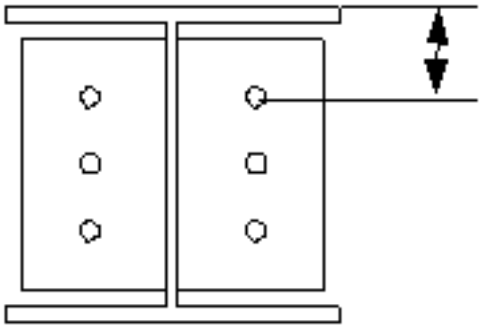
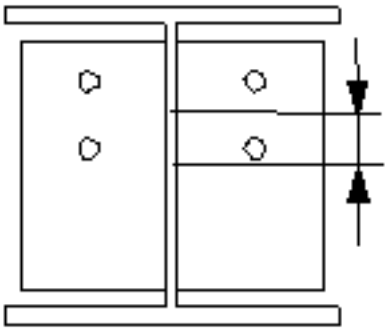
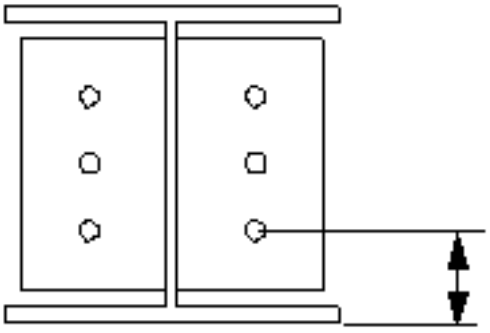
Bolts tab

Use the **Bolts** tab to control the bolt group dimensions.

Bolt group dimensions



	Description
1	Dimension for vertical bolt group position.

	Description
2	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

	Description
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Number of rows of bolts through the main part.

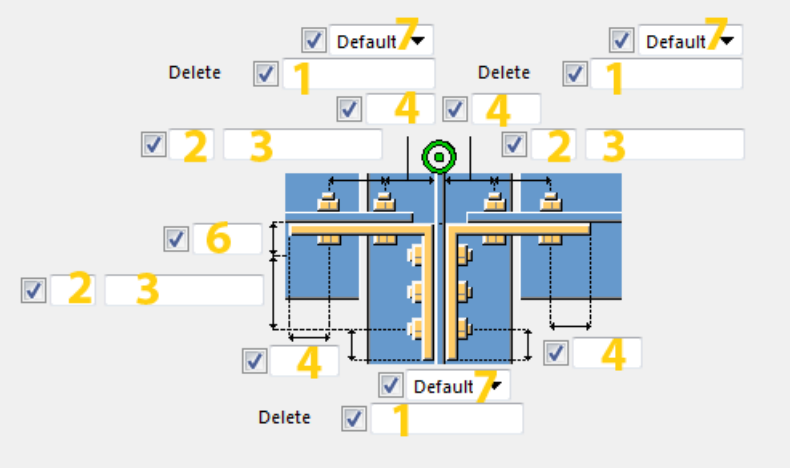
Staggering of bolts on clip angle

Option for second secondary	Option for first secondary	Description
		Default Bolts are not staggered. AutoDefaults can change this option.
		Bolts are not staggered. The bolts that connect the clip angle to the secondary part are on the same horizontal level as the bolts that connect the clip angle to the main part.
		Bolts on the main part are staggered. The bolts that connect the clip angle to the main part are moved downwards by half the bolt vertical spacing value.
		Bolts on the secondary part are staggered. The bolts that connect the clip angle to the secondary part are moved downwards by half the bolt vertical spacing value.
		Bolts on the secondary part are staggered. The bolts that connect the clip angle to the sloped secondary part are parallel to the secondary part.

Attachment type

Option	Description
	Default Both parts are bolted. AutoDefaults can change this option.
	Automatic When the main part is a tube profile, the clip angles are welded to the main part and bolted to the secondary part. Otherwise the clip angles are bolted to both parts.
	Both parts are bolted.
	Main part is welded and secondary part is bolted.
	Main part is bolted and secondary part is welded.
	Both parts are welded.
	Main part is not bolted.
	Secondary part is not welded.
	Secondary part is not bolted.
	Both parts are bolted and welded.

Bolt group dimensions



	Description
1	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Bolt edge distance.
5	Location where the bolts should be attached.
6	Angle leg length.
7	Location where the bolts should be attached: **Site** / **Workshop**.

Bolt settings tab

Use the **Bolt settings** tab to control the bolts and welds that connect the clip angle to the main part and to the secondary part.

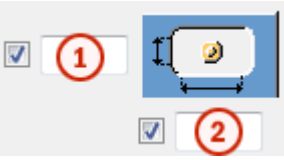
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.

Option	Description	Default
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

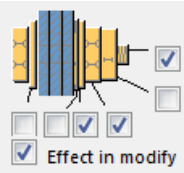
Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Same bolt length for all

Set this option to **Yes** to have the same bolt length for all bolts. The longest bolt defines how long the shorter bolts are with extra length.

Use different settings for secondary parts

Set to **Yes** to define different settings for the second secondary part. The default is that the same settings are used for both secondary parts.

Number of clip angle welds

Define the number of welds that connect the clip angle to the main part and/or the secondary part.

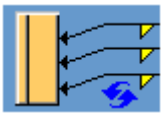
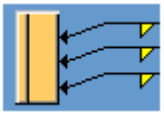
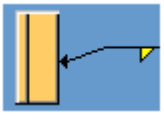
Option	Description
	Default Three welds are created to the clip angle. AutoDefaults can change this option.
	Three welds are created to the clip angle.
	One weld is created to the clip angle.

Plate washers tab

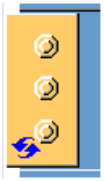
Use the **Plate washers** tab to define the plate washer properties to the main part and to the secondary parts.

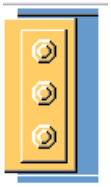
Option	Description
Plate washer	Plate washer thickness, width and height.

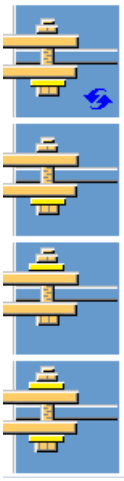
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

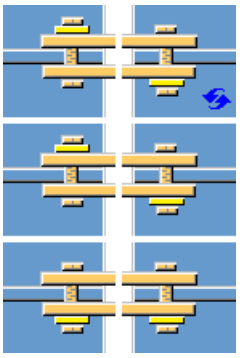
Plate washer

Define plate washers for bolts and select the plate washer side.

Option	Description
	Default No plate washer AutoDefaults can change this option.
	No plate washer

Option	Description
	One plate washer
	Individual square plate washers for each bolt
	Individual round plate washers for each bolt

Option	Description
	Select whether the plate washer is created for one clip angle or both clip angles.

Option	Description
	Select whether the plate washers are placed symmetrically or asymmetrically.

Angle box tab

Use the **Angle box** tab to add a seat angle.

Seat angle

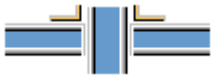
The purpose of seat angles is to carry loads from the secondary part. Seat angles can be positioned to top, bottom or both flanges of the secondary part. The seat angle can be stiffened, and bolted or welded to the main and secondary parts.

Option	Description
Stiffeners	Stiffener thickness, width and height.
Top angle NS, **Bottom angle NS**, **Top angle FS**, **Bottom angle FS**	Select the seat angle profile by selecting it from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Top seat angle position

Option	Description
	Default No seat angle is created. AutoDefaults can change this option.
	No seat angle is created.

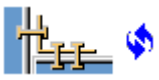
Option	Description
	Seat angle is created at the top of the second secondary part flange.
	Seat angle is created at the top of the first secondary part flange.
	Seat angles are created at the top of both flanges.

Bottom seat angle position

Option	Description
	Default No seat angle is created. AutoDefaults can change this option.
	No seat angle is created.
	Seat angle is created at the bottom of the second secondary part flange
	Seat angle is created at the bottom of the second secondary part flange.
	Seat angles are created at the bottom of both flanges.

Seat angle attachment

Seat angle is positioned at the top or at the bottom of the secondary part.

Option for top seat angle	Option for bottom seat angle	Description
		Default Bolted Seat angle is bolted to the main part and to the secondary part. AutoDefaults can change this option.
		Bolted Seat angle is bolted to the main part and to the secondary part.

Option for top seat angle	Option for bottom seat angle	Description
		Welded-bolted Seat angle is welded to the main part and bolted to the secondary part.
		Bolted-welded Seat angle is bolted to the main part and welded to the secondary part.
		Welded Seat angle is welded to the main part and to the secondary part.

Stiffener type

Option	Description
	Default Rectangular stiffener plate AutoDefaults can change this option.
	Rectangular stiffener plate
	Triangular stiffener plate
	The line connecting the ends of the seat angle legs defines the stiffener plate shape.

Seat angle orientation

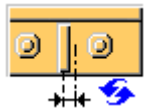
Option	Description
	Default The longer leg of the seat angle is connected to the secondary part. AutoDefaults can change this option.
	The longer leg of the seat angle is connected to the secondary part.

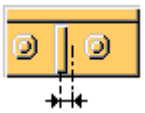
Option	Description
	The longer leg of the seat angle is connected to the main part.
	Automatic The longer leg of the seat angle is connected to the part where bolts reach furthest from the seat angle corner.

Seat angle rotation

Option	Description
	Default Seat angle is not rotated. AutoDefaults can change this option.
	Seat angle is not rotated.
	Seat angle is rotated horizontally 90 degrees. To stiffen the rotated seat angle, select the **Middle stiffeners** option in the **Middle stiffener position** list.

Middle stiffener position

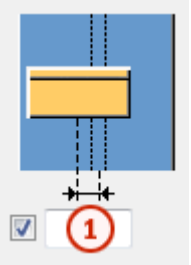
Option	Description
	Default According to bolts AutoDefaults can change this option.
	No middle stiffener plate is created.
	Middle stiffeners The stiffener plate is positioned in the middle of the seat angle.

Option	Description
	Enter the number of middle stiffeners in the **Number of middle stiffeners** box. Multiple stiffeners are centered and equally spaced.
	According to bolts The stiffener plate is positioned between the bolts in the middle of the bolt spacing. By default, stiffener is created between every two bolts. Enter the number of middle stiffeners in the box below the **According to bolts** option.

Side stiffener position

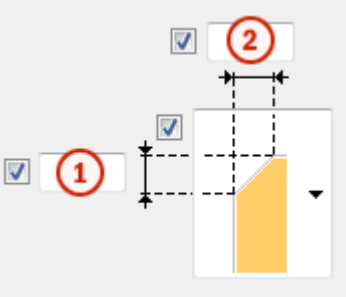
Option	Description
	Default No side stiffeners are created. AutoDefaults can change this option.
	No side stiffeners are created.
	Near side stiffeners are created.
	Far side stiffeners are created.
	Near side and far side stiffeners are created.

Seat angle offset



	Description
1	Seat angle horizontal offset from the center line of the main part.

Chamfer dimensions



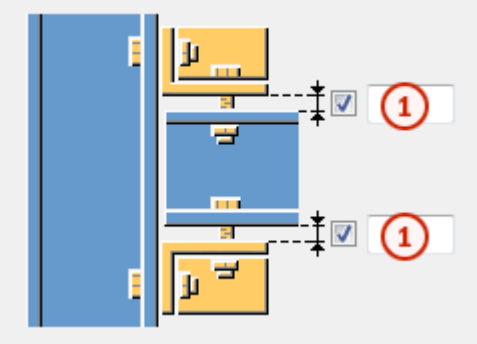
	Description
1	Vertical dimension of the chamfer.
2	Horizontal dimension of the chamfer.

Chamfer type

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer

Option	Description
	Concave arc chamfer

Gap

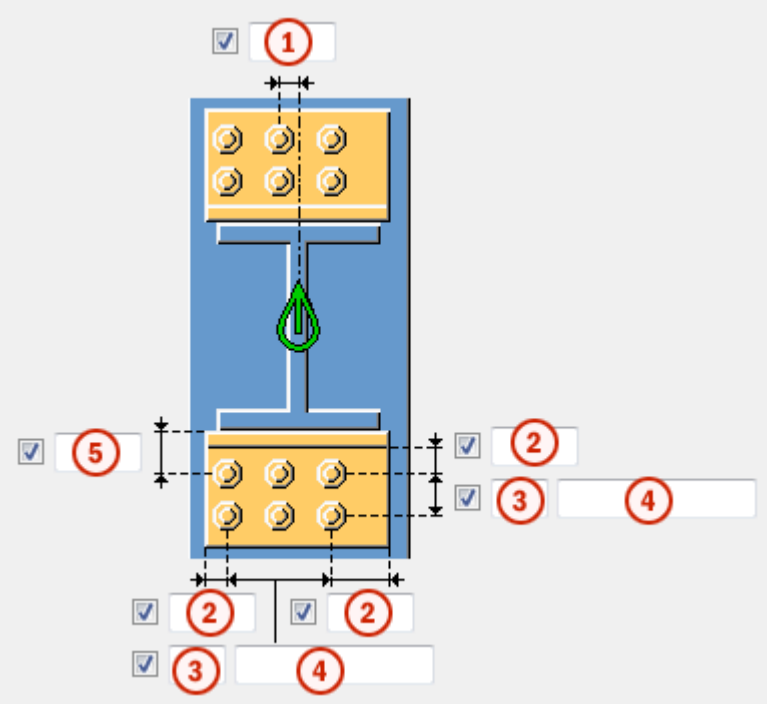


	Description
1	Top gap and bottom gap between the seat angle and the secondary part.

BoxPBolts tab

Use the **BoxPBolts** tab to control properties of the bolts that connect the seat angle to the main part.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position. The dimension is defined from the middle line of the secondary beam.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position. The dimension is defined from the bottom of the secondary beam.

Top

Top refers to the bolt group that connects the top seat angle to the main part.

Bottom

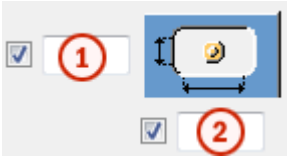
Bottom refers to the bolt group that connects the bottom seat angle to the main part.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	

Option	Description	Default
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

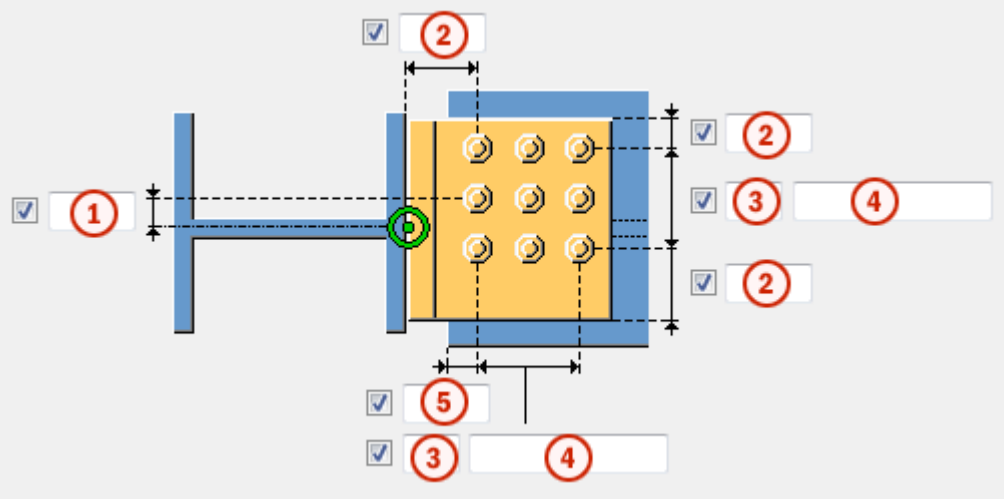
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

BoxSBolts tab

Use the **BoxSBolts** tab to control properties of the bolts that connect the seat angle to the secondary part.

Bolt group dimension



	Description
1	Dimension for vertical bolt group position. The dimension is defined from the middle line of the secondary beam.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.

	Description
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for horizontal bolt group position. The dimension is defined from the bottom of the secondary beam.

Top

Top refers to the bolt group that connects the top seat angle to the secondary part.

Bottom

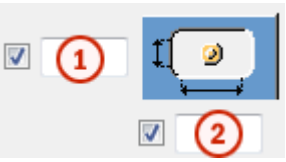
Bottom refers to the bolt group that connects the bottom seat angle to the secondary part.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Beam cut tab

Use the **Beam cut** tab to control weld backing bars, weld access holes, beam end preparations, and flange cuts.

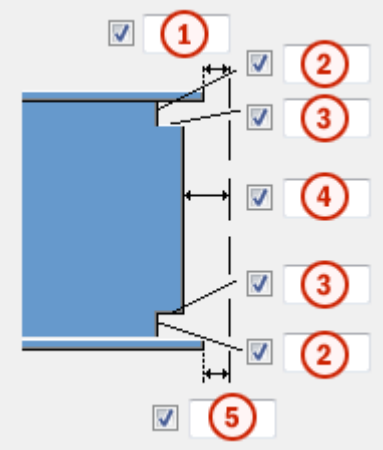
Weld backing bar

Option	Description
Weld backing bar	Weld backing bar thickness and width.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

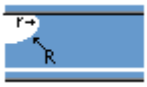
Option	Description	Default
	assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Weld access hole dimensions



	Description
1	Gap between the secondary part top flange and the main part.
2	Vertical dimensions for the top and the bottom weld access holes.
3	Horizontal dimensions for the top and the bottom weld access holes.
4	Gap between the secondary part web and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.
5	Gap between the secondary part bottom flange and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.

Weld access holes

Option	Description	Default
	Default Round weld access hole AutoDefaults can change this option.	
	Round weld access hole	
	Square weld access hole	
	Diagonal weld access hole	
	Round weld access hole with a radius that you can define in r ☑	
	Extended cone-shaped weld access hole with a radius and dimensions that you can define in R ☑ and Top Prep x ☑ Bottom Prep x ☑	
	Cone-shaped weld access hole with radiuses that you can define in R ☑ and r ☑ Capital **R** defines the large radius (height). Small **r** defines the small radius.	R = 35 r = 10

Beam end preparation

Option	Description
	Default Top and bottom flange are prepared. AutoDefaults can change this option.
	Automatic Top and bottom flange are prepared.
	Beam end is not prepared.
	Top and bottom flange are prepared.
	Top flange is prepared.
	Bottom flange is prepared.

Flange cut

Option for top flange	Option for bottom flange	Description
		Default Flange is not cut. AutoDefaults can change this option.
		Flange is not cut.
		Flange is cut.

Weld backing bars

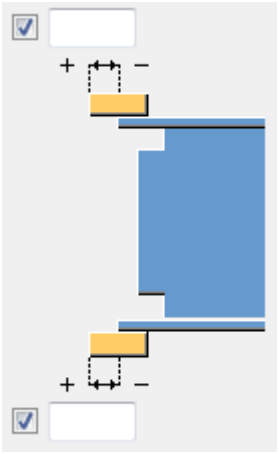
Option for top backing bar	Option for bottom backing bar	Description
		Default Backing bars are created inside the flanges. AutoDefaults can change this option.
		No backing bars are created.
		Backing bars are created inside the flanges.
		Backing bars are created outside the flanges.

Weld backing bar length

Enter the length of the weld backing bar in the box below the options.

Option	Description
	Default Absolute length of the backing bar AutoDefaults can change this option.
	Absolute length of the backing bar
	Extension beyond the edge of the flange

Weld backing bar position

Option	Description
	Enter a positive or a negative value to move the front end of the backing bar relative to the end of the flange.

Assembly type

Define the location where the weld backing bar welds are made. When you select the **Workshop** option, Tekla Structures includes the backing bars in the assembly.

General tab

Click the links below to find out more:

General tab

Design type tab

Click the links below to find out more:

Design type tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

5.3 Bent plate connections

This section introduces bent plate connection components available in Tekla Structures.

Click the links below to find out more:

- Bent plate (151) (page 1285)
- Bent plate (190) (page 1310)

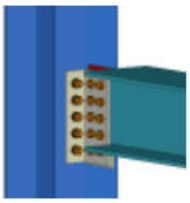
Bent plate (151)

Bent plate (151) connects a beam to a column or a beam using one or two bolted or welded bent plates. The end of the beam can be cut square or bevel to accommodate a sloping secondary beam. By default, notches are created to the secondary beam flanges when needed. You can define on which side of the secondary beam web the bent plate is placed in single plate connections. Double plate connections can also be created. Haunch plates are created welded to the secondary beam flanges when required.

Objects created

- Bent plates
- Haunch plates
- Stiffeners
- Bolts
- Cuts
- Welds

Use for

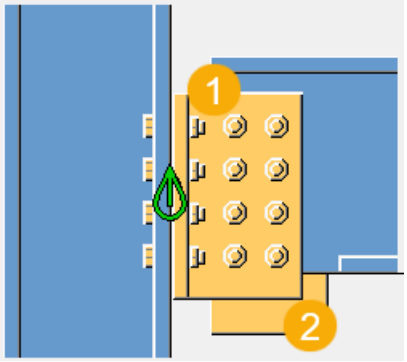
Situation	Description
	Beam connected to a column using a bent plate.

Selection order

1. Select the main part (column or beam).
2. Select the secondary part.

   The connection is created automatically when the secondary part is selected.

Part identification key

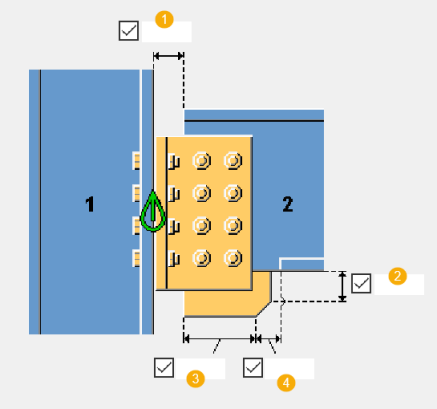


	Description
1	Bent plate
2	Haunch plate

Picture tab

Use the **Picture** tab to define the connection dimensions.

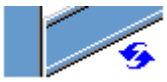
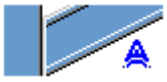
Dimensions



	Description
1	Distance between the main part and the secondary part (beam). By default, the distance is `20` mm.
2	Haunch plate height to the beginning of the chamfer.
3	Haunch plate width to the beginning of the chamfer.
4	Distance from the bent plate edge to the lower flange notch. You can define this dimension if the plate extends below the bottom of the secondary beam.

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.
	Square Cuts the end of the secondary beam square.
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.

Beam flange cut

Define how the secondary beam flange end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the flange bevel.
	Square Cuts a part of the flange square and leaves a part of it bevel.

Haunch plate creation

Options	Description
	Default Haunch plates are not created. AutoDefaults can change this option.

Options	Description
	Haunch plates are not created.
	Top and bottom haunch plates are created.
	Top haunch plate is created.
	Bottom haunch plate is created.

Parts tab

Use the **Parts** tab to define the bent plate properties and positions.

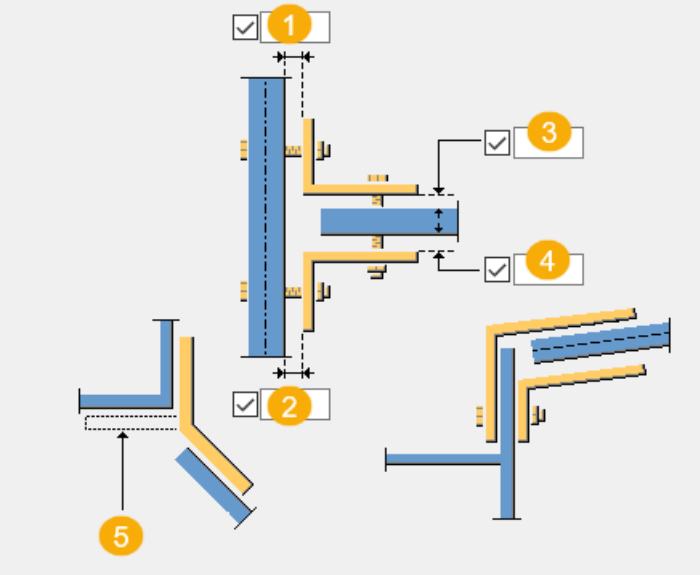
Parts

Option	Description
Bent plate NS	Thickness of the near side bent plate.
Bent plate FS	Thickness of the far side bent plate.
Top Web Ext	Thickness, width and height of the top web extension.
Btm Web Ext	Thickness, width and height of the bottom web extension.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Dimensions



	Description
1	Gap between the right bent plate and the main part.
2	Gap between the left bent plate and the main part.
3	Gap between the right bent plate and the secondary part.
4	Gap between the left bent plate and the secondary part.
5	Collision plane change.

Bent plate position

	Default Creates two bent plates, near side and far side. AutoDefaults can change this option.
	Automatic Near side and far side bent plates are selected automatically.
	Near side bent plate

	Far side bent plate
	Far side switched Creates a far side bent plate with a leg pointing to the near side.
	Near side switched Creates a far side bent plate with a leg pointing to the far side.
	Creates two bent plates, near side and far side.

Attachment to the main part

Option	Description
	Default The bent plate is bolted to the main part. AutoDefaults can change this option.
	Automatic The bent plate is bolted to the main part.
	The bent plate is bolted to the main part.
	The bent plate is welded to the main part.
	The bent plate is bolted and welded to the main part.

Attachment to the secondary part

Option	Description
	Default The bent plate is bolted to the secondary part. AutoDefaults can change this option.
	Automatic The bent plate is bolted to the secondary part.
	The bent plate is bolted to the secondary part.

Option	Description
	The bent plate is welded to the secondary part.
	The bent plate is bolted and welded to the secondary part.

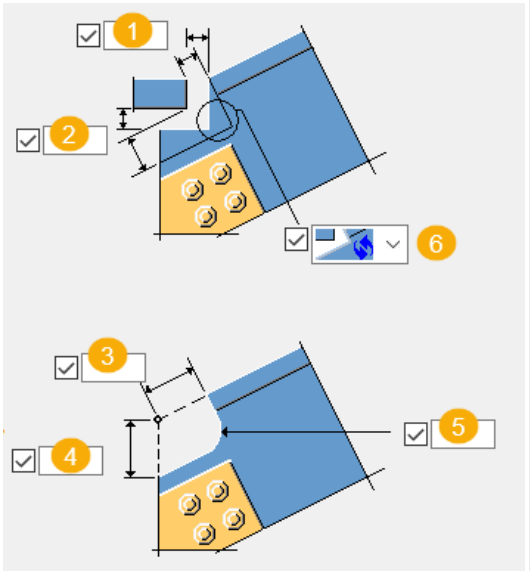
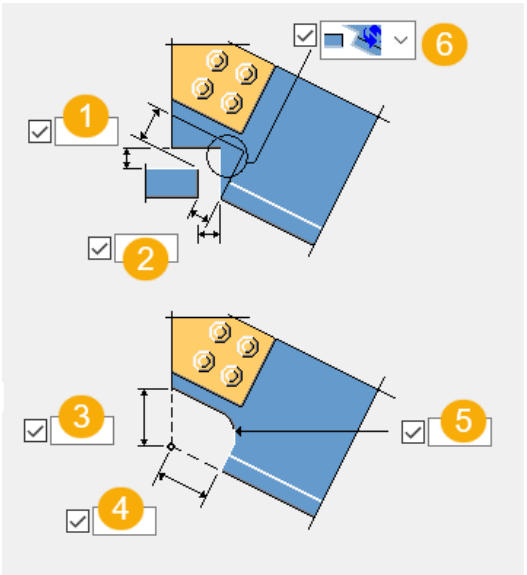
Attachment point control

Option	Description
	Default The collision plane is not changed from the default position. AutoDefaults can change this option.
	The collision plane is not changed from the default position.
	The collision plane is changed to the nearest surface.
	The collision plane is changed to the farthest surface.

Top notch / Btm notch tab

Use the **Top notch** and **Btm notch** tabs to define the notch settings and dimensions.

Notch dimensions

Option	Option
	

	Description
1	Horizontal dimension for the notch clearing. The distance is measured from the intersection point of three planes: top or bottom of the flange plane, end of the flange plane, and the inner web or side surface, depending on the type of clearance selected.
2	Vertical dimension for the notch clearing. The distance is measured from the intersection point of three planes: bottom of the flange plane, end of the flange plane, and the inner web or side surface, depending on the type of clearance selected.
3	Horizontal dimension for the notch override. The distance is measured from the intersection point of three planes: top of the flange plane, fitting plane, and the inner web or side surface, depending on the type of clearance selected.
4	Vertical dimension for the notch override. The distance is measured from the intersection point of three planes: top of the flange plane, fitting plane, and the inner web or side surface, depending on the type of clearance selected.
5	Radius of the notch cut.
6	Vertical notch orientation.

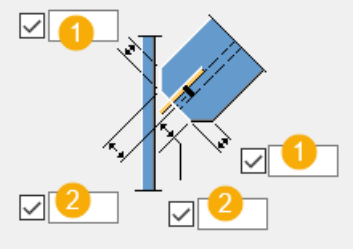
Notch settings

Do notch	• **Yes** enables notching. • **No** disables notching, but does not disable stripping and flange coping. Use the **Strip / Cope** options for disabling stripping and flange coping.
Type of clearance	• **Side with flange** Use the main part flange and the inner side plane of the secondary part as a reference for the clearing. • **Web with plate** Use the inner shear tab plane and the inner web plane of the secondary part as a reference for the clearing.

	• **Web with flange** Use the flange of the main part and the inner web plane of the secondary part as a reference for the clearing.
Strip/Cope option	• **Automatic strip** Based on the shear tab shape and edges, the connection strips the secondary part flanges as necessary. • **Never strip or cope** Stripping or coping is not done. • **Force strip** Connection always strips, even when the shear tab edges are not clashing or close to the secondary part flanges. Set the notch cut length in **Forcing Dist** and the depth in **Force Depth**. • **Automatic flange cope** Based on the shear tab shape and edges, the connection copes the secondary part flanges as necessary. • **Force flange cope** The connection always copes, even when the shear tab edges are not clashing or close to the secondary part flanges. Set the notch cut length in **Forcing Dist** and the depth in **Force Depth**. • **Strip also other side** Both secondary part flanges are stripped.

Flange cut options

Define the flange cut dimensions and the cut thickness.



	Description
1	Dimension for additional flange cut. The dimension is measured from the end of the beam if not fitted, or from the fitting surface if fitted.
2	Dimension for additional flange cut. The dimension is measured from the center line of the beam.

Vertical notch orientation

	Bottom	
		Default Square to the secondary part. AutoDefaults can change this option.
		Automatic Square to the secondary part.
		Square to the secondary part. The notch is cut vertically square relative to the secondary part.
		Square to main part The notch is cut vertically square relative to the main part.

Notch orientation

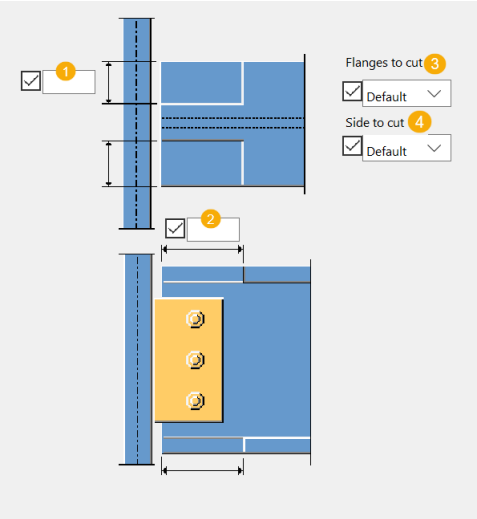
Option	Description
	Default Square to the main part. AutoDefaults can change this option.

Option	Description
	Automatic Square to the main part.
	Square to the secondary part. The notch is cut horizontally square relative to the secondary part.
	Square to the main part. The notch is cut horizontally square relative to the main part.

Flange cuts tab

Use the **Flange cuts** tab to define how the flanges are cut.

Dimensions



	Description
1	Transversal dimension
2	Longitudinal dimension
3	• **Top**: Top flange is cut. • **Bottom**: Bottom flange is cut. • **Both**: Both the top and the bottom flange are cut.
4	• **Near**: Near side is cut. • **Far**: Far side is cut. • **Both**: Both the near and far side are cut.

Stiffeners tab

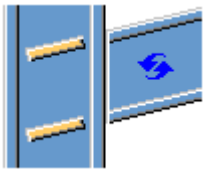
Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position, and type.

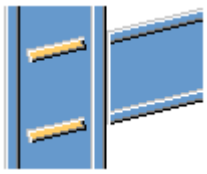
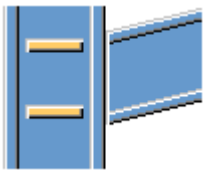
Stiffener plate dimensions

Option	Description
Top NS	Thickness, width, and height of the top near side stiffener.
Top FS	Thickness, width, and height of the top far side stiffener.
Bottom NS	Thickness, width, and height of the bottom near side stiffener.
Bottom FS	Thickness, width, and height of the bottom far side stiffener.

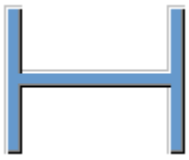
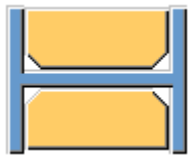
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Stiffener orientation

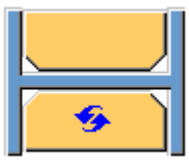
Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.

Option	Description
	Stiffeners are parallel to the secondary part.
	Stiffeners are perpendicular to the main part.

Stiffener creation

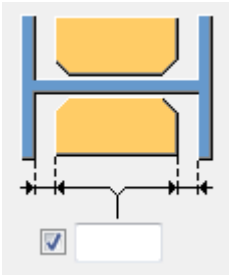
Option	Description
	Default Stiffeners are created. AutoDefaults can change this option.
	Automatic Stiffeners are created when necessary.
	No stiffeners are created.
	Stiffeners are created.

Stiffener shape

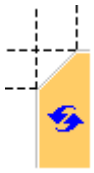
Option	Description
	Default Line chamfered stiffener plates AutoDefaults can change this option.

Option	Description
	Automatic Line chamfered stiffener plates
	Square stiffener plates Stiffener plates with a gap for the main part web rounding
	Line chamfered stiffener plates

Stiffener gap

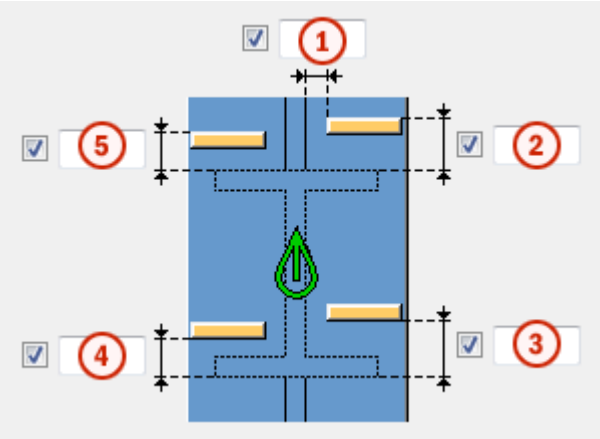
Option	Description
	Size of the gap between the flanges and the stiffener.

Chamfer type

Option	Description
	Default. Line chamfer AutoDefaults can change this option.
	No chamfer

Option	Description
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Stiffener positions

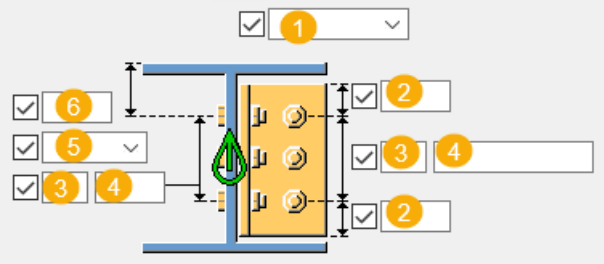


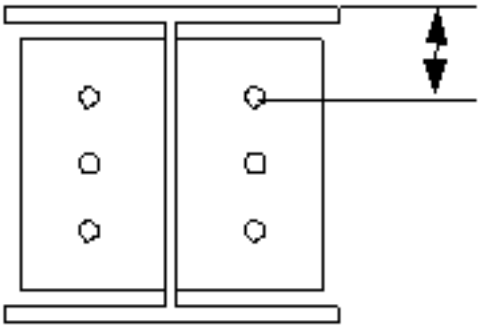
	Description
1	Size of the gap between the stiffener and the beam web edge.
2	Size of the gap between the top near side stiffener and the beam flange edge.
3	Size of the gap between the bottom near side stiffener and the beam flange edge.
4	Size of the gap between the bottom far side stiffener and the beam flange edge.
5	Size of the gap between the top far side stiffener and the beam flange edge.

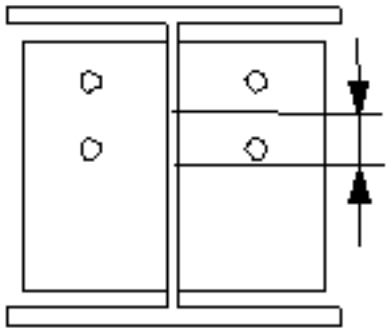
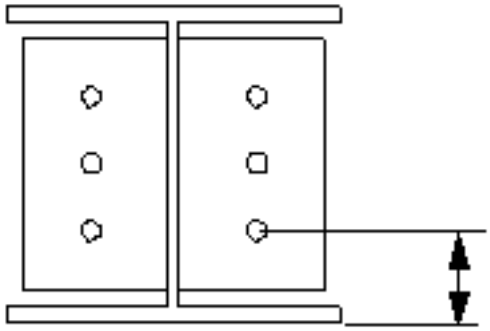
Bolts tab

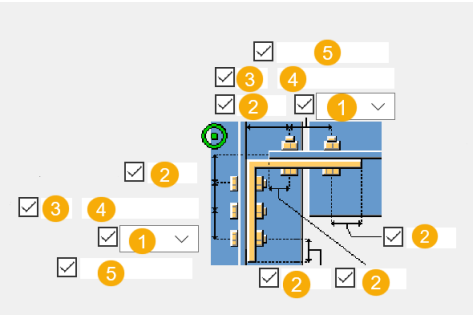
Use the **Bolts** tab to control the bolt group dimensions and bolt properties.

Bolt group dimensions



	Description
1	Select main bolted part: beam or bent plate.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
6	Bolt edge distance. Edge distance from the center of the bolt to the top of the beam.



	Description
1	Select the location where the bolts should be attached.

	Description
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Bolt alignment

Main bolt alignment	Secondary bolt alignment	Description
		Default Square to the secondary part. AutoDefaults can change this option.
		Automatic Square to the secondary part.
		Square to the secondary part.
		Square to the main part.
		Vertically staggered.
		Horizontally staggered.

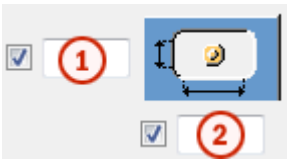
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.

Option	Description	Default
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.

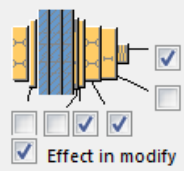


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Staggering of bolts on clip angle

Option	Description
	Default Bolts are not staggered. AutoDefaults can change this option.

Option	Description
	Bolts are not staggered. The bolts that connect the clip angle to the secondary part are on the same horizontal level as the bolts that connect the clip angle to the main part.
	Bolts on the main part are staggered. The bolts that connect the clip angle to the main part are moved downwards by half the bolt vertical spacing value.
	Bolts on the secondary part are staggered. The bolts that connect the clip angle to the secondary part are moved downwards by half the bolt vertical spacing value.
	Bolts on the secondary part are staggered. The bolts that connect the clip angle to the sloped secondary part are parallel to the secondary part.

Beam cut tab

Use the **Beam cut** tab to control weld backing bars, weld access holes, beam end preparations, and flange cuts.

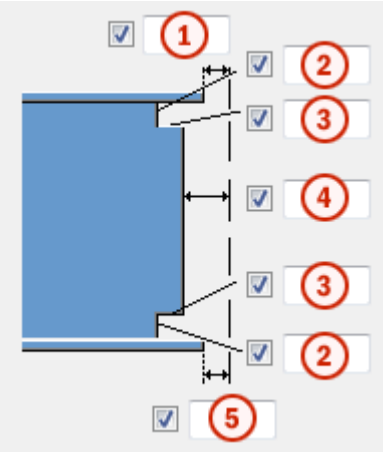
Weld backing bar

Option	Description
Backing bar	Thickness and width of the weld backing bar.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Option	Description	Default
Finish	Describes how the part surface has been treated.	

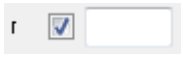
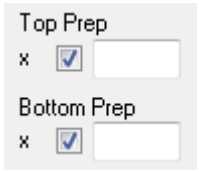
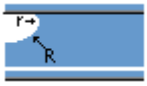
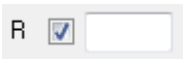
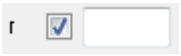
Weld access hole dimensions



	Description
1	Gap between the secondary part top flange and the main part.
2	Vertical dimensions for the top and the bottom weld access holes.
3	Horizontal dimensions for the top and the bottom weld access holes.
4	Gap between the secondary part web and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.
5	Gap between the secondary part bottom flange and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.

Weld access holes

Option	Description	Default
	Default Round weld access hole AutoDefaults can change this option.	
	Round weld access hole	

Option	Description	Default
	Square weld access hole	
	Diagonal weld access hole	
	Round weld access hole with a radius that you can define in 	
	Extended cone-shaped weld access hole with a radius and dimensions that you can define in and 	
	Cone-shaped weld access hole with radiuses that you can define in  and  Capital **R** defines the large radius (height). Small **r** defines the small radius.	`R = 35` `r = 10`

Beam end preparation

Option	Description
	Default Top and bottom flange are prepared. AutoDefaults can change this option.
	Automatic Top and bottom flange are prepared.

Option	Description
	Beam end is not prepared.
	Top and bottom flange are prepared.
	Top flange is prepared.
	Bottom flange is prepared.

Flange cut

Option for top flange	Option for bottom flange	Description
		Default Flange is not cut. AutoDefaults can change this option.
		Flange is not cut.
		Flange is cut.

Weld backing bars

Option for top backing bar	Option for bottom backing bar	Description
		Default Backing bars are created inside the flanges. AutoDefaults can change this option.
		No backing bars are created.
		Backing bars are created inside the flanges.

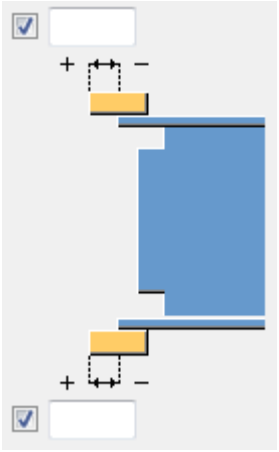
Option for top backing bar	Option for bottom backing bar	Description
		Backing bars are created outside the flanges.

Weld backing bar length

Enter the length of the weld backing bar in the box below the options.

Option	Description
	Default Absolute length of the backing bar AutoDefaults can change this option.
	Absolute length of the backing bar
	Extension beyond the edge of the flange

Weld backing bar position

Option	Description
	Enter a positive or a negative value to move the front end of the backing bar relative to the end of the flange.

Assembly type

Define the location where the weld backing bar welds are made. When you select the **Workshop** option, Tekla Structures includes the backing bars in the assembly.

General tab

Click the link below to find out more:

General tab

Design type tab

Click the link below to find out more:

Design type tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

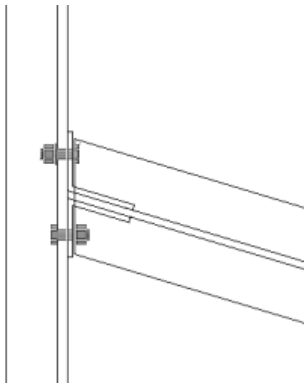
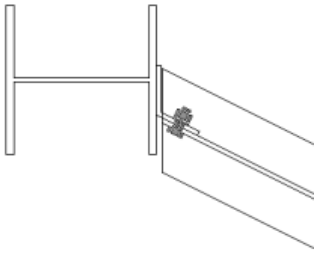
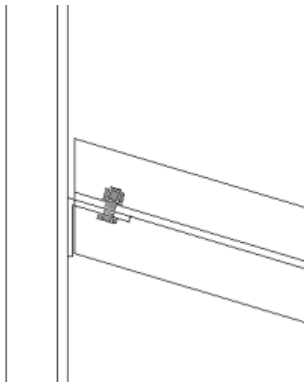
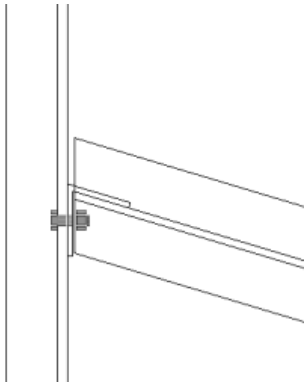
Bent plate (190)

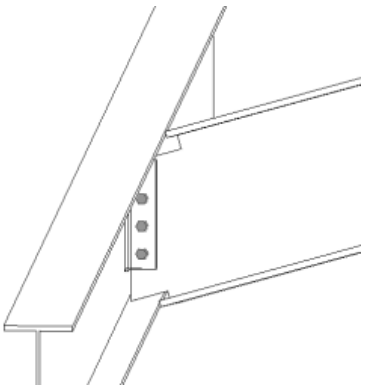
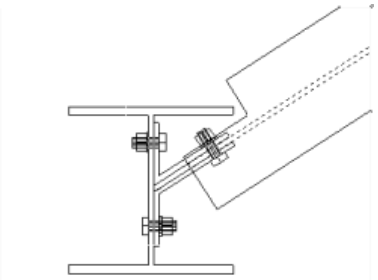
Bent plate (190) connects two beams or a beam and a column using one or two bolted or welded bent plates. The secondary beam can be leveled or sloped and/or skewed. Welded haunch plates are optional.

Objects created

- Bent plate (1 or 2)
- Stiffeners (optional)
- Top and bottom haunch plate (optional)
- Weld backing bars (optional)
- Bolts
- Welds
- Cuts

Use for

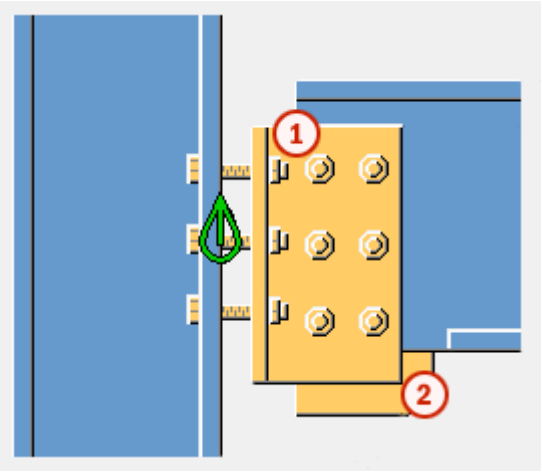
Situation	Description
	Bent plate connecting two beams. The secondary part is skewed.
	Bent plate connected to a column flange. The secondary part is skewed.
	Bent plate connecting two beams. The secondary part is skewed.
	Bent plate connecting two beams. The secondary part is skewed. The plate can be placed to various locations.

Situation	Description
	Bent plate connecting two beams. The secondary part is skewed and sloped.
	Bent plate connected to a column web. The secondary part is skewed.

Selection order

1. Select the main part (beam or column).
2. Select the secondary part (beam). The connection is created automatically when the secondary part is selected.

Part identification key



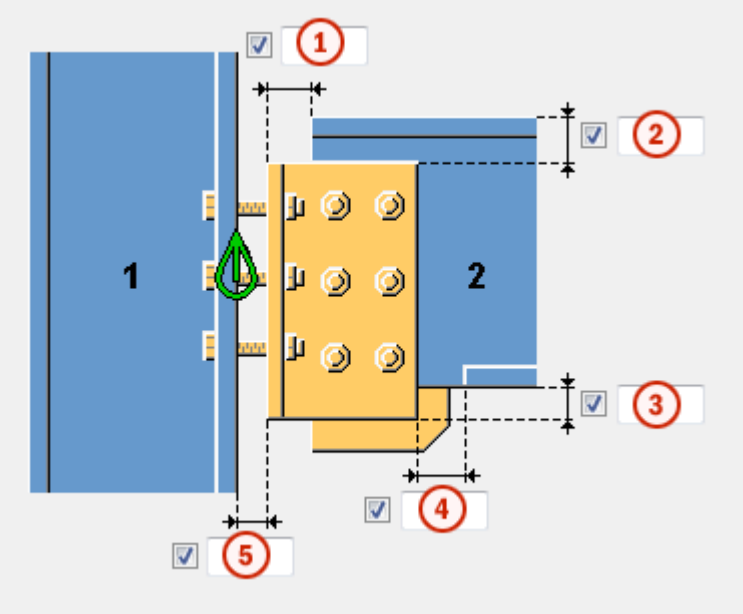
	Part
1	Bent plate
2	Haunch plate

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to control the position of the bent plate and to define how the beam end is cut.

Dimensions

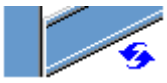
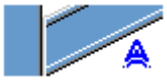


	Description	Default
1	Cut length for the secondary part. The cutting point is defined from the bent plate edge.	`20 mm`
2	Bent plate upper edge distance from the top of the secondary part. The upper edge position of the plate modifies the bent plate height. Positive value moves the top position closer to the beam center and thus decreases the bent plate size. Negative value increases the bent plate size.	If no value is entered, bolts and bolt edge distances define the size of the plate.

	Description	Default
3	Bent plate lower edge distance from the bottom of the secondary part. The lower edge position of the plate modifies the bent plate height. Positive value moves the bottom position closer to the beam center and thus decreases the plate size. Negative values increase the plate size.	If no value is entered, bolts and bolt edge distances define the size of the plate.
4	Size of the strip made to the secondary part flange. The cutting point of the flange is defined from the bent plate edge.	The flange is automatically stripped when the bent plate crosses the flange. 10 mm
5	Gap between the main part and the bent plate.	0

Beam end cut

Define how the beam end is cut. The secondary part is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Automatic If the secondary part slopes less than 10 degrees, a square cut is created. Otherwise, a bevel cut is made to the end of the secondary part.
	Square Creates a square cut to the end of the secondary part.
	Bevel Cuts the end of the secondary part parallel to the edge of the main part.

Plates tab

Use the **Plates** tab to control the thickness, position and attachment of the bent plate. The dimensions on the **Picture** tab and the **Bolts** tab also affect the size of the bent plate.

Bent plate NS/FS

Option	Description
Bent plate	Bent plate thickness.

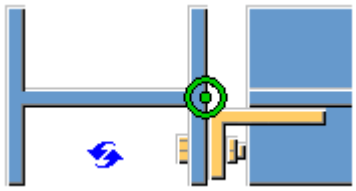
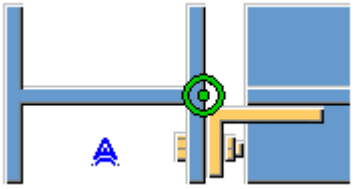
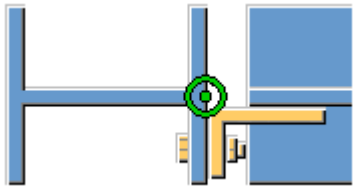
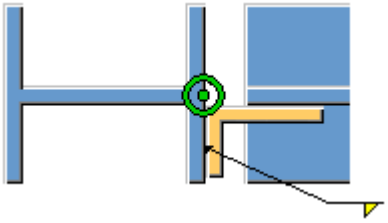
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

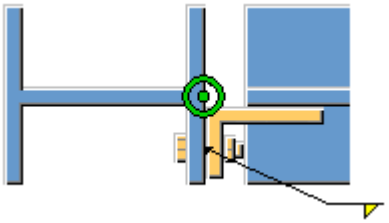
Bent plate position

Option	Description
	Default Near side and far side bent plate AutoDefaults can change this option.
	Automatic Near side and far side bent plates are created automatically.
	Near side switched Creates a near side bent plate with a leg pointing to the far side.
	Far side switched Creates a far side bent plate with a leg pointing to the near side.

Option	Description
	Far side
	Near side
	Near side and far side

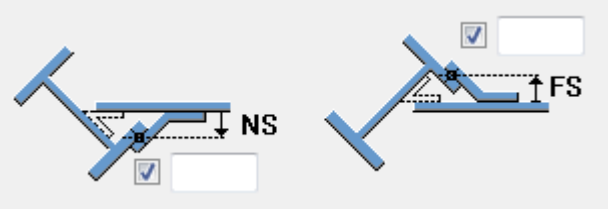
Main and secondary part attachment type

Option	Description
	Default Bent plate is bolted to the main part. AutoDefaults can change this option.
	Automatic Bolted
	Bolted
	Welded

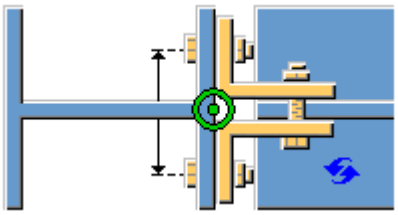
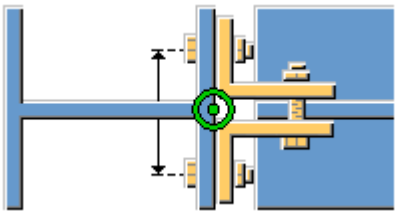
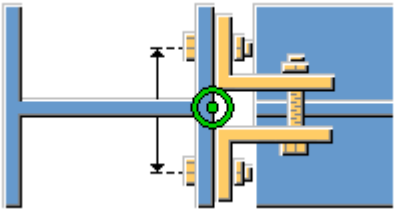
Option	Description
	Bolted and welded

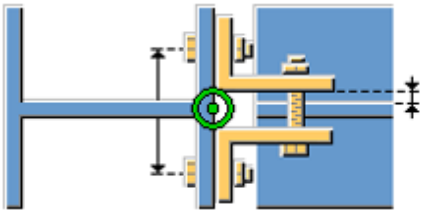
Attachment point control

Define whether the bent plate is attached to the web or to the flange of the main part.



Bolt spacing and weld gap

Option	Description
	Default Define the bolt spacing. Weld gap is not created. AutoDefaults can change this option.
	Define the bolt spacing. Weld gap is not created.
	Define the bolt spacing. Weld gap is created.

Option	Description
	Define the bolt spacing and the weld gap.

Stiffeners tab

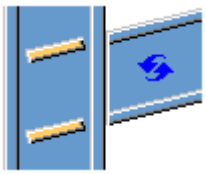
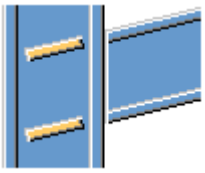
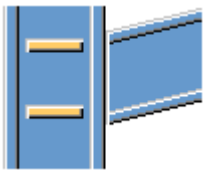
Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position, and type.

Stiffener plate dimensions

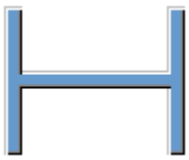
Option	Description
Top NS	Top near side stiffener thickness, width and height.
Top FS	Top far side stiffener thickness, width and height.
Bottom NS	Bottom near side stiffener thickness, width and height.
Bottom FS	Bottom far side stiffener thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Stiffener orientation

Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are parallel to the secondary part.
	Stiffeners are perpendicular to the main part.

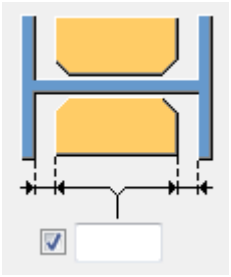
Stiffener creation

Option	Description
	Default Stiffeners are created. AutoDefaults can change this option.
	Automatic Stiffeners are created when necessary.
	No stiffeners are created.
	Stiffeners are created.

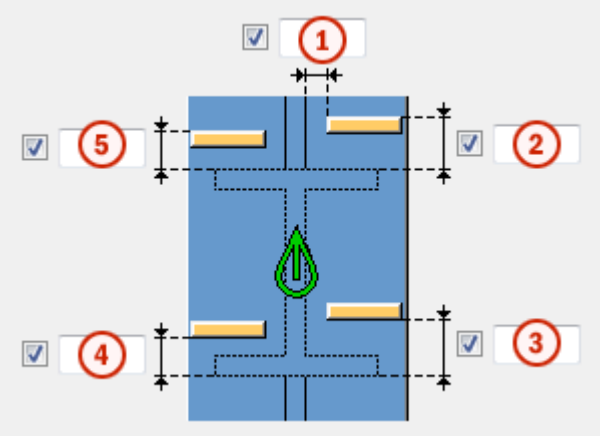
Stiffener shape

Option	Description
	Default Line chamfered stiffener plates AutoDefaults can change this option.
	Automatic Line chamfered stiffener plates
	Square stiffener plates Stiffener plates with a gap for the main part web rounding
	Line chamfered stiffener plates

Stiffener gap

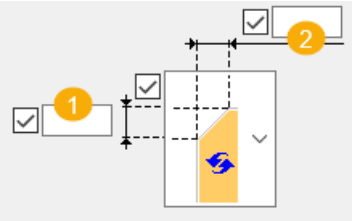
Option	Description
	Size of the gap between the flanges and the stiffener.

Stiffener positions



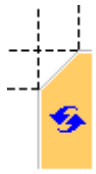
	Description
1	Size of the gap between the stiffener and the beam web edge.
2	Size of the gap between the top near side stiffener and the beam flange edge.
3	Size of the gap between the bottom near side stiffener and the beam flange edge.
4	Size of the gap between the bottom far side stiffener and the beam flange edge.
5	Size of the gap between the top far side stiffener and the beam flange edge.

Chamfer dimensions



	Description	Default
1	Vertical dimension of the chamfer.	`10 mm`
2	Horizontal dimension of the chamfer.	`10 mm`

Chamfer type

Option	Description
	Default. Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Haunch tab

Use the **Haunch** tab to control the haunch plate creation and chamfers in the secondary beam flanges.

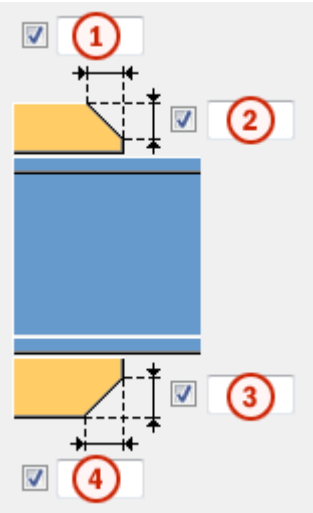
Haunch plates

Option	Description
Top plate	Top haunch plate thickness, width and height.
Bottom plate	Bottom haunch plate thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

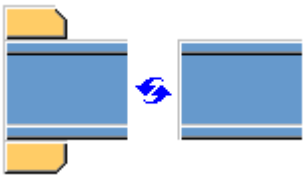
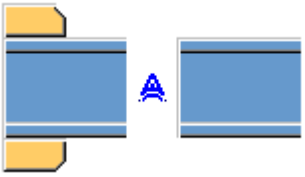
Option	Description	Default
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Haunch plate chamfers



	Description
1	Width of the top haunch plate chamfer.
2	Height of the top haunch plate chamfer.
3	Height of the bottom haunch plate chamfer.
4	Width of the bottom haunch plate chamfer.

Haunch plate creation

Option	Description
	Default Top and bottom haunch plates are created, if needed. AutoDefaults can change this option.
	Automatic Top or bottom haunch plate or both are created, if needed.

Option	Description
	Top and bottom haunch plates are created. To create a single plate, enter `0` in the thickness (**t**) field for the plate you do not need (top or bottom plate).
	Haunch plates are not created.

Notch tab

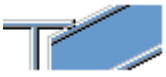
Use the **Notch** tab to automatically create notches for the secondary beams and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

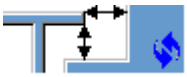
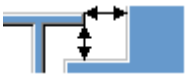
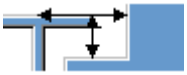
Automatic notching options affect both the top and the bottom flange.

Notch shape

Automatic notching is switched on when you select a notch shape.

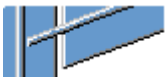
Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

Enter the horizontal and vertical values for the cuts.

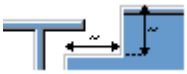
Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.

Option	Description
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.

Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.

Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

Side of flange notch

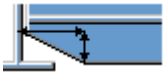
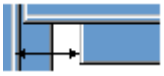
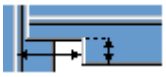
The side of flange notch defines on which side of the beam the notches are created.

Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

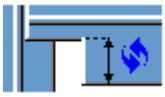
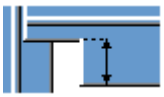
Flange notch shape

The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.

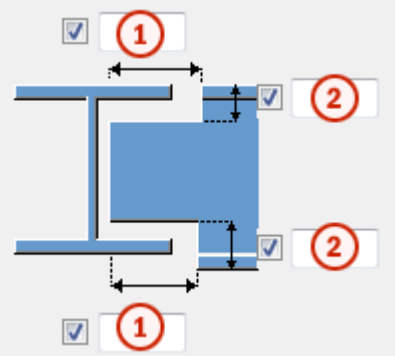
Option	Description
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.

Cut dimensions

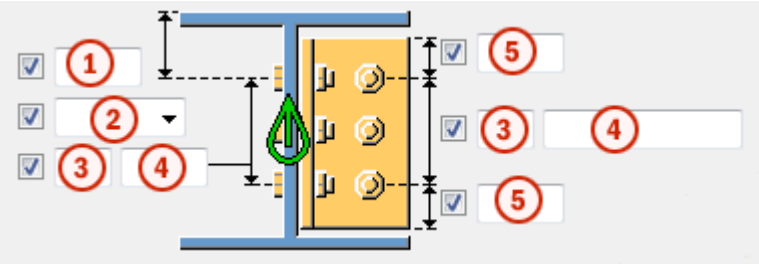


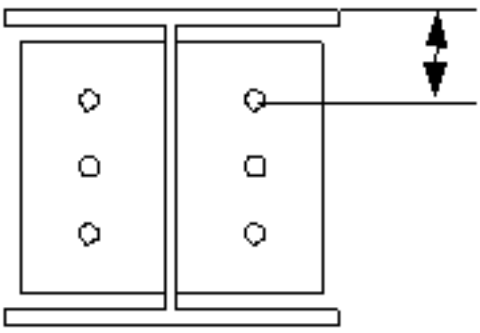
	Description	Default
1	Dimensions for the horizontal flange cuts.	10 mm
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest 5 mm.

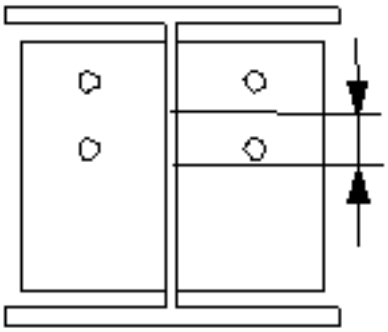
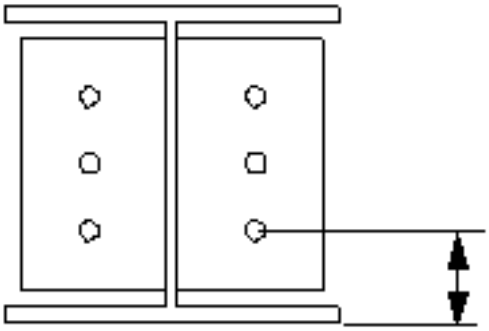
Bolts tab

Use the **Bolts** tab to control the properties of the bolts that connect the bent plate to the main part and to the secondary part.

Bolt group dimensions



	Description
1	Dimension for vertical bolt group position.
2	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

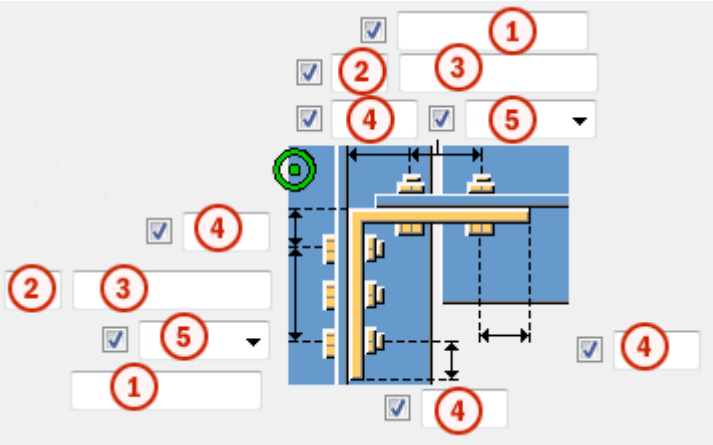
	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.

Staggering of bolts on bent plate

Option	Description
	Default Bolts are not staggered. AutoDefaults can change this option.

Option	Description
	Bolts are not staggered. The bolts that connect the bent plate to the secondary part are on the same horizontal level as the bolts that connect the bent plate to the main part.
	Bolts on the main part are staggered. The bolts that connect the bent plate to the main part are moved downwards by half the bolt vertical spacing value.
	Bolts on the secondary part are staggered. The bolts that connect the bent plate to the secondary part are moved downwards by half the bolt vertical spacing value.
	Bolts on the secondary part are staggered. The bolts that connect the bent plate to the sloped secondary part are parallel to the secondary part.

Bolt group dimensions



	Description
1	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

	Description
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Bolt edge distance.
5	Location where the bolts should be attached.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

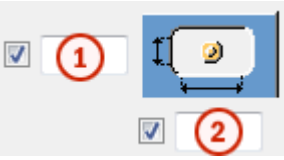
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

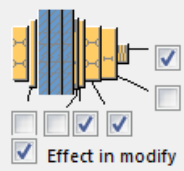


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Beam cut tab

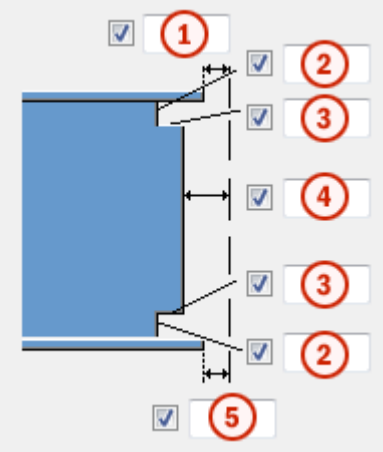
Use the **Beam cut** tab to control weld backing bars, weld access holes, beam end preparations, and flange cuts.

Weld backing bar

Option	Description
Weld backing bar	Weld backing bar thickness and width.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

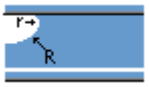
Weld access hole dimensions



	Description
1	Gap between the secondary part top flange and the main part.
2	Vertical dimensions for the top and the bottom weld access holes.
3	Horizontal dimensions for the top and the bottom weld access holes.
4	Gap between the secondary part web and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.
5	Gap between the secondary part bottom flange and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.

Weld access holes

Option	Description	Default
	Default Round weld access hole AutoDefaults can change this option.	
	Round weld access hole	
	Square weld access hole	
	Diagonal weld access hole	

Option	Description	Default
	Round weld access hole with a radius that you can define in r ☑	
	Extended cone-shaped weld access hole with a radius and dimensions that you can define in R ☑ and Top Prep x ☑ Bottom Prep x ☑	
	Cone-shaped weld access hole with radiuses that you can define in R ☑ and r ☑ Capital **R** defines the large radius (height). Small **r** defines the small radius.	R = 35 r = 10

Beam end preparation

Option	Description
	Default Top and bottom flange are prepared. AutoDefaults can change this option.
	Automatic Top and bottom flange are prepared.
	Beam end is not prepared.
	Top and bottom flange are prepared.

Option	Description
	Top flange is prepared.
	Bottom flange is prepared.

Flange cut

Option for top flange	Option for bottom flange	Description
		Default Flange is not cut. AutoDefaults can change this option.
		Flange is not cut.
		Flange is cut.

Weld backing bars

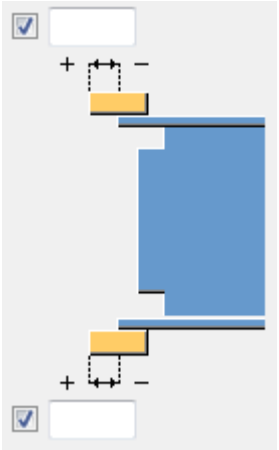
Option for top backing bar	Option for bottom backing bar	Description
		Default Backing bars are created inside the flanges. AutoDefaults can change this option.
		No backing bars are created.
		Backing bars are created inside the flanges.
		Backing bars are created outside the flanges.

Weld backing bar length

Enter the length of the weld backing bar in the box below the options.

Option	Description
	Default Absolute length of the backing bar AutoDefaults can change this option.
	Absolute length of the backing bar
	Extension beyond the edge of the flange

Weld backing bar position

Option	Description
	Enter a positive or a negative value to move the front end of the backing bar relative to the end of the flange.

Assembly type

Define the location where the weld backing bar welds are made. When you select the **Workshop** option, Tekla Structures includes the backing bars in the assembly.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

5.4 End plate connections and details

This section introduces components that can be used in end plate connection and details.

- Seating cap plate (2) (page 1340)
- Bearing plate (7) (page 1348)
- Column - 2 beams (14) (page 1356)
- Joining plates (14) (page 1373)
- Wooden purlin shoe (15) (page 1399)
- Two sided end plate (24) (page 1387)
- Stiffened end plate (27) (page 1409)
- End plate (29) (page 1426)
- Seating cap (37) (page 1441)
- Haunch (40) (page 1457)
- Cranked beam (41) (page 1482)
- Partial stiff end plate (65) (page 1495)
- End plate (101) (page 1514)
- End plate with compensating flange plates (111) (page 1524)
- Two-sided end plate with compensating flange plates (112) (page 1535)
- Two sided end plate (115) (page 1547)
- Stub connection (119) (page 1559)
- Two sided end plate (142) (page 1570)
- End plate (144) (page 1600)
- End plate detail (1002) (page 1632)
- Angle cut (1057) (page 1634)
- Cast-in plate (1069) (page 1637)

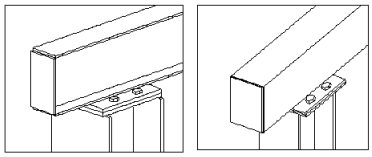
Seating cap plate (2)

Seating cap plate (2) connects a beam to the column head. The connection creates two plates, one welded to the bottom of the beam and one to the column head. The plates are bolted together.

Objects created

- Cap plates
- End plate
- Bolts
- Welds

Use for

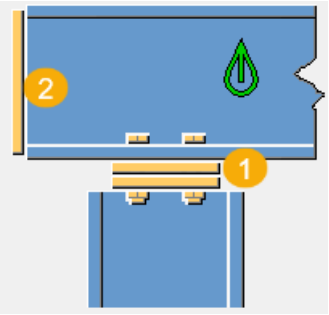
Situation	Description
	Beam and column connected with two cap plates that are bolted together. The end plate is created to the beam end.

Selection order

1. Select the main part (beam or column).
2. Select the secondary part (beam or column).

   The connection is created automatically when the secondary part is selected.

Part identification key

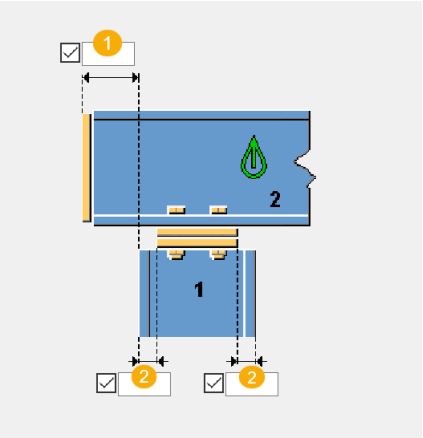


	Description
1	Upper cap plate Lower cap plate
2	End plate

Picture tab

Use the **Picture** tab to define the connection dimensions.

Dimensions



	Description	Default
1	Cut distance of the secondary beam end from the main part edge.	
2	End plate edge distance from the main part edge.	`-10 mm` Positive values move the end plate edge towards the column centre.

Parts tab

Use the **Parts** tab to define the part properties.

Parts

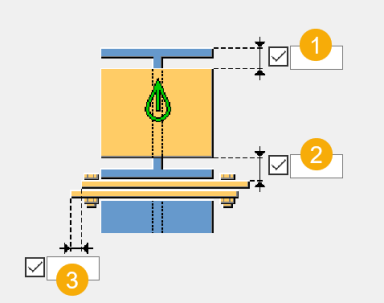
Option	Description
Lower cap plate	Thickness, width and height of the cap plate.
Upper cap plate	Thickness, width and height of the cap plate.
End plate	Thickness, width and height of the end plate. If you leave the plate thickness empty, the end plate is not created.

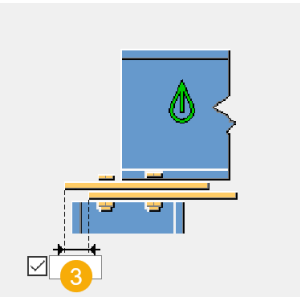
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

Use the **Parameters** tab to define the plate dimensions.

Plate dimensions

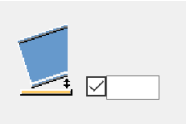




	Description
1	End plate edge distance to the secondary part flange. This distance affects the size of the end plate welded to the secondary part end.

	Description
2	End plate edge distance to the upper cap plate. This distance affects the size of the end plate welded to the secondary part end.
3	Cap plate edge dimension.

Gap size



Define the limit value for the gap between the end plate and the secondary beam. Use this when the beam is slightly curved or sloped to decide if the end angle is so small that the beam end can be straight.

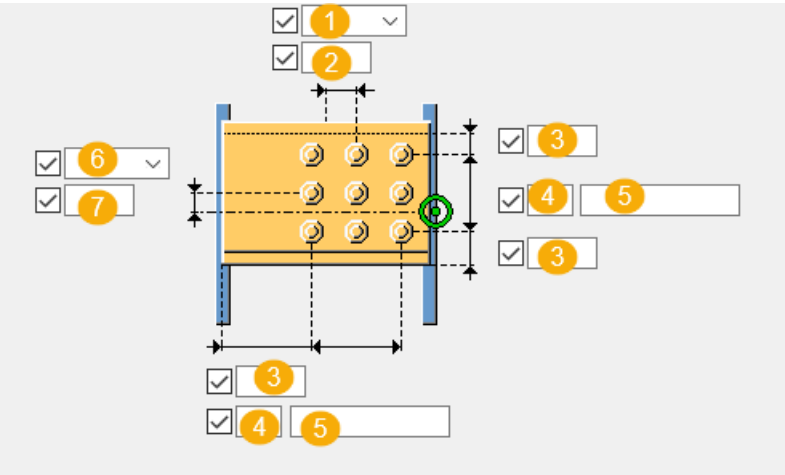
If the actual gap is smaller than this value, the end of the beam is left straight.

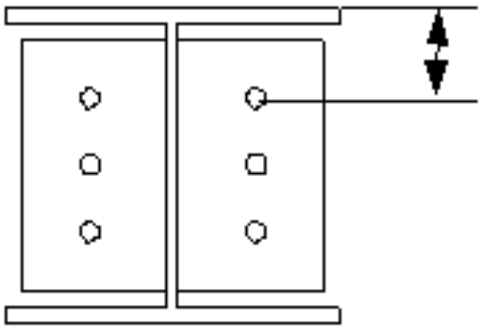
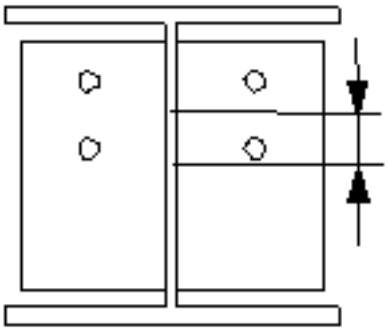
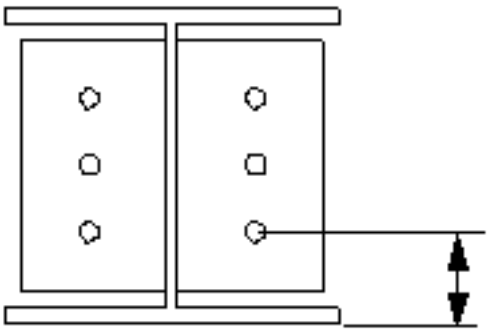
If the actual gap is larger than this value, the end of the beam is fitted to the end plate.

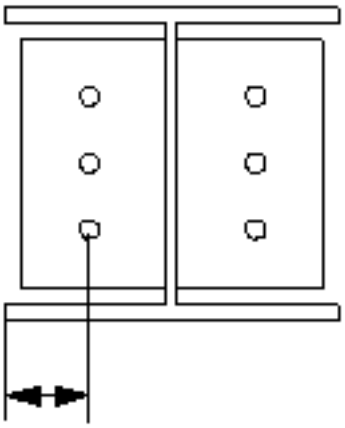
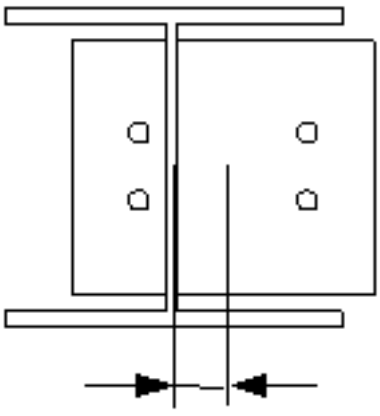
Bolts tab

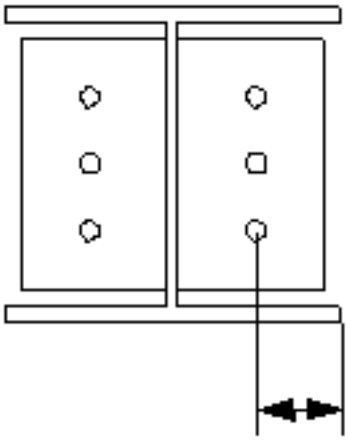
Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
2	Dimension for vertical bolt group position.

	Description
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt.  • **Middle**: From the center line of the secondary part to the center line of the bolts. 

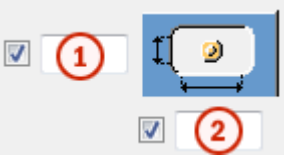
	Description
	• **Right**: From the right edge of the secondary part to the rightmost bolt. 
7	Dimension for horizontal bolt group position.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

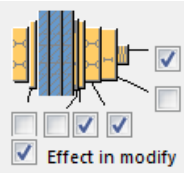


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

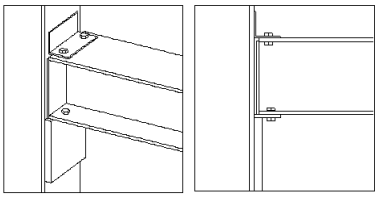
Bearing plate (7)

Bearing plate (7) connects a beam to a column. A console seat plate welded to the column carries the shear forces. The beam upper flange is bolted to an L profile. The lower flange is bolted to a shim plate.

Objects created

- Console seat plate
- End plate
- Shim plate
- Clip angle (L profile)
- Bolts
- Welds

Use for

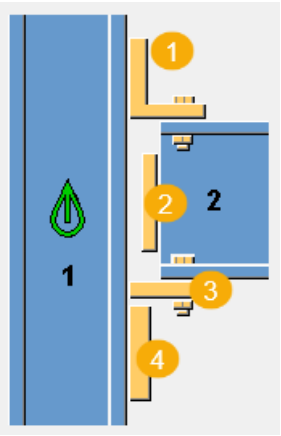
Option	Description
	Beam is connected to a column. The clip angle, shim plate, and console are created.

Selection order

1. Select the main part (column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

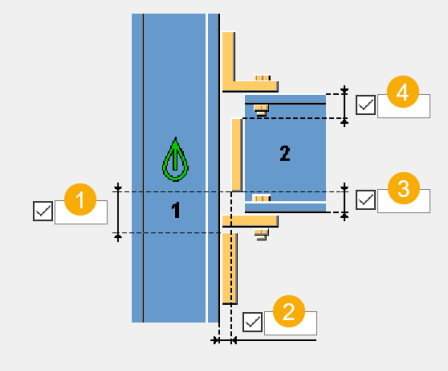


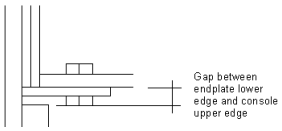
	Description
1	Clip angle (L profile)
2	End plate
3	Shim plate
	Console seat plate

Picture tab

Use the **Picture** tab to define the connection dimensions.

Dimensions



	Description	Default
1	Console seat edge distance from the end plate lower edge.  	
2	Gap between the end plate and the column.	10 mm
3	End plate edge distance from the beam lower edge. Positive values move the end plate lower edge upward and negative values downward.	End plate lower edge is set to the same position as the beam lower edge.
4	End plate edge distance from the beam upper edge. Positive values move the end plate upper edge downward and negative values upward.	10 mm

Parts tab

Use the **Parts** tab to define the part properties.

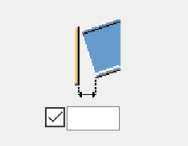
Parts

Option	Description	
Seat	Thickness, width, and height of the console seat.	Thickness: `30 mm` If the width is left empty, the secondary beam width is used. If the height is left empty, the secondary beam height is used.
End plate	Thickness and width of the end plate.	Thickness: Equal to the secondary part web thickness (rounded up to thickness 8, 10, 12, 16, 20, 25, 30, 35, 40, 45, and so on). If the width is left empty, the secondary beam width is used.
Fitting plate	Thickness of the shim plate.	`10 mm`
L profile	Select the profile from the profile catalog.	`L100-6`
L seat	Select the profile from the profile catalog. An angle console can be created under the end plate. When an L console is used, the shim plate and the console seat plate are not created.	

Parameters tab

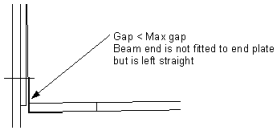
Use the **Parameters** to control the end plate gap size.

Gap size



Define the limit value for the gap between the end plate and the secondary beam. Use this when the beam is slightly curved or sloped to decide if the end angle is so small that the beam end can be straight.

If the actual gap is smaller than this value, the end of the beam is left straight.

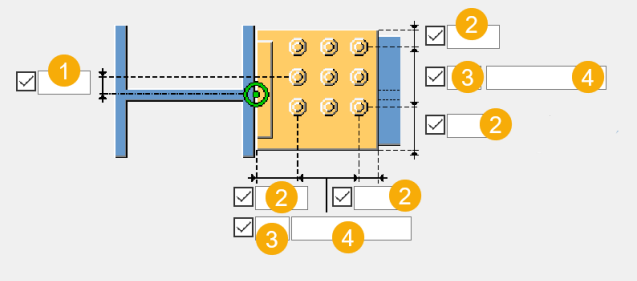


If the actual gap is larger than this value, the end of the beam is fitted to the end plate.

Console Blt tab

Use the **Console Blt** tab to define the bolt group dimensions and bolt properties of the console seat plate.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

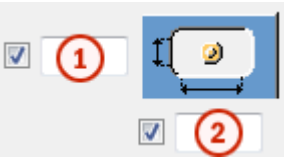
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.

Option	Description	Default
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

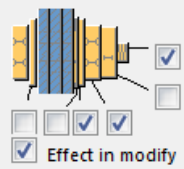


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

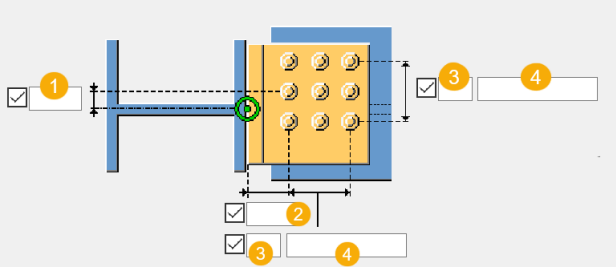
Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

L-profile Blt tab

Use the **L-profile Blt** tab to define the bolt group dimensions and bolt properties of the L profile bolts.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

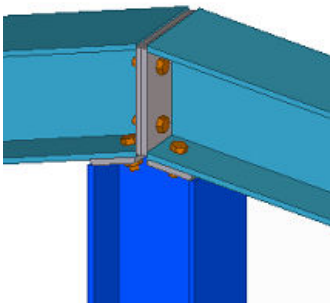
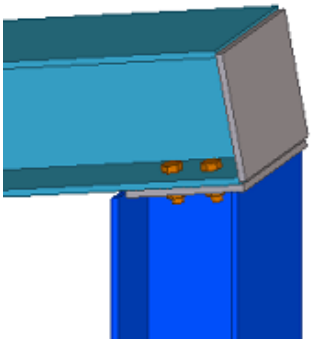
Column - 2 beams (14)

Column - 2 beams (14) connects two beams to a column. The beam ends rest on the column. The secondary beams can be horizontal or sloped.

Objects created

- End plates
- Cap plates
- Bolts
- Welds

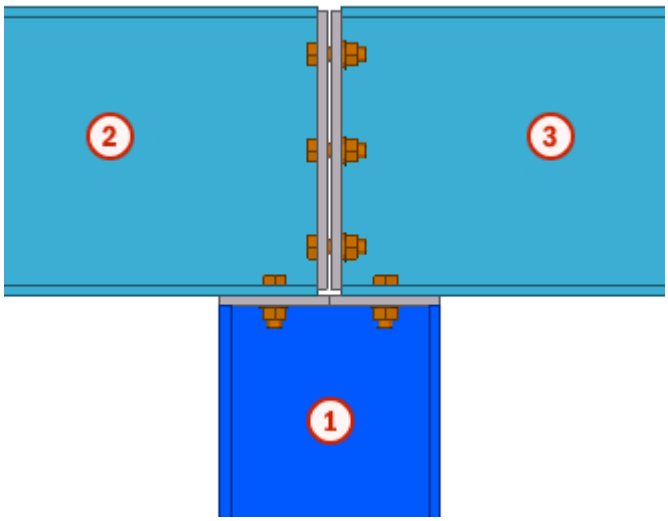
Use for

Situation	Description
	Connection between a column and two beams.
	Connection between a column and a beam. You can use the component with only one secondary beam but then the settings are more difficult to define, and there are fewer situations where the component can be used. If you use the component with only one secondary beam, enter a negative value for the gap between end plates.

Selection order

1. Select the main part (column).
2. Select the first secondary part (beam).
3. Select the second secondary part (beam).
4. Click the middle mouse button to create the connection.

Part identification key



	Part
1	Column
2	Beam
3	Beam

Picture tab

Use the **Picture** tab to control the plate sizes.

Plate 1, 2, 3, 4

Plates 1 and 2 are the end plates on the beams. Plates 3 and 4 are the cap plates between the columns and the beams.

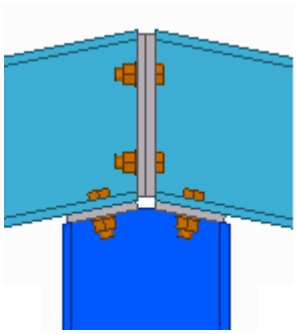
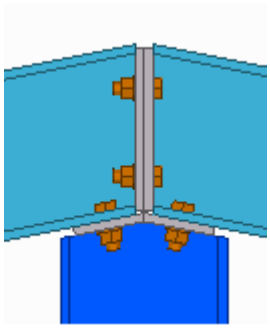
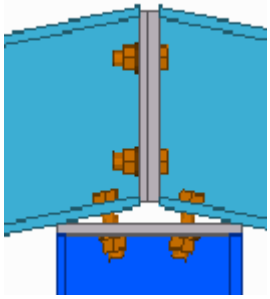
Option	Description
Plate 1, **2**, **3**, **4**	Plate thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Plate 3 + 4

Define the type of the connection between the column and beams.

Option	Description	Example
2 plates	Two plates are created. The column is chamfered twice.	
Folded plate	A folded plate is created. The plate can be unfolded on a workshop drawing. The column is chamfered twice.	
1 plate	One plate is created. The plate is always oriented horizontally. The column is fitted perpendicularly.	

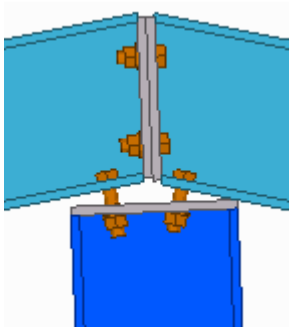
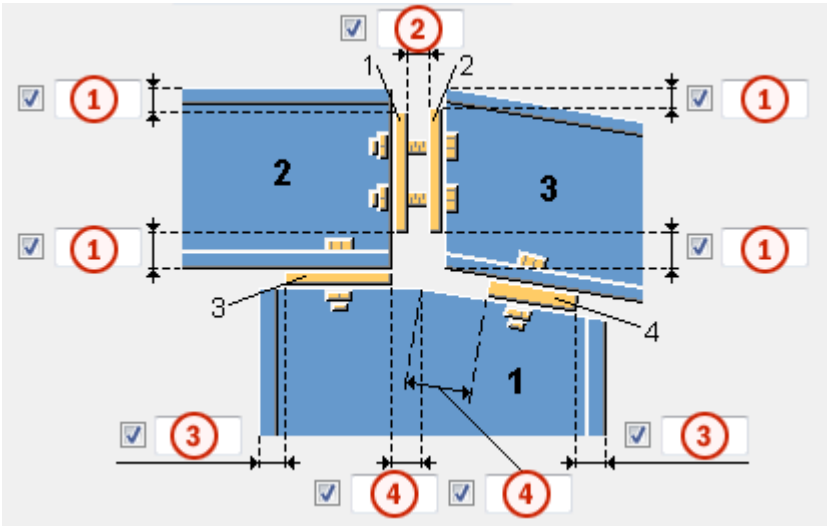
Option	Description	Example
Plate perp to column	One plate is created. The column is fitted perpendicularly, even if the column is positioned in a skewed direction.	

Plate distances

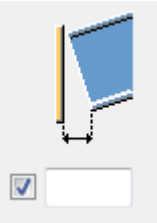


	Description	Default
1	Plate distance from the beam edge.	5 mm
2	Gap between the end plates. If you use the component with only one secondary beam, enter a negative value.	
3	Plate distance from the column edges.	
4	Plate distance from the column center.	

End plates

Select how the end plates are positioned.

Gap size



Define the limit value for the gap between the end plate and the beam, or for the gap between the cap plate and the column. Use this when the beam or the column is slightly curved or sloped to decide if the end angle is so small that the end can be straight.

If the actual gap is smaller than this value, the end of the beam is left straight.

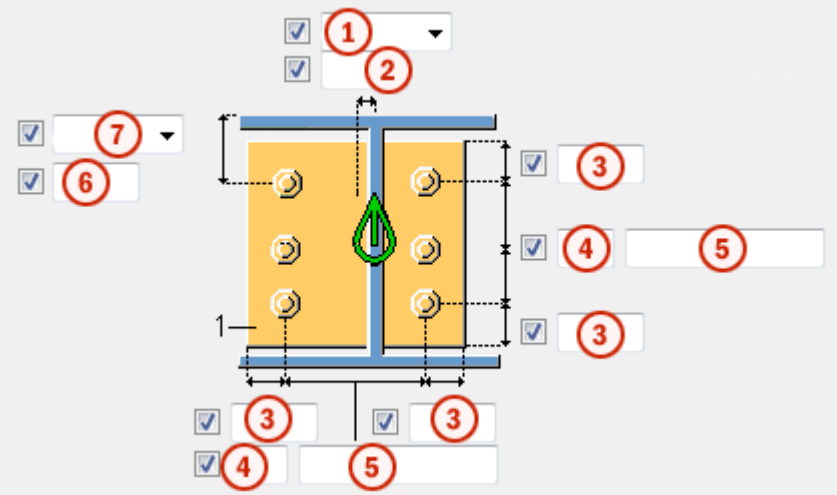
If the actual gap is larger than this value, the end is fitted to the end plate or to the cap plate.

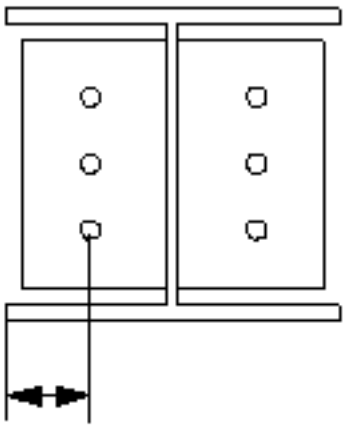
The default value is `5 mm`.

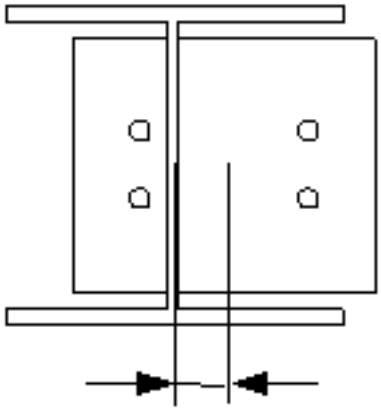
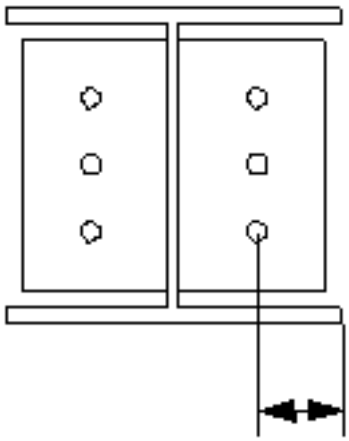
Bolts 1 - 2 tab

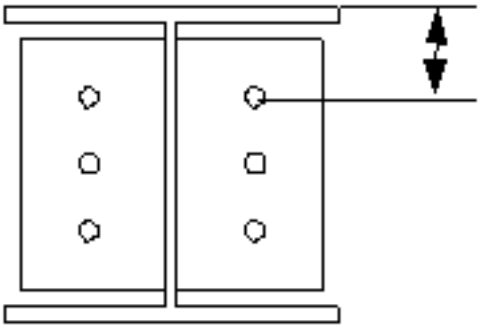
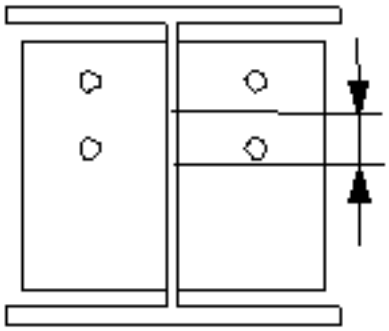
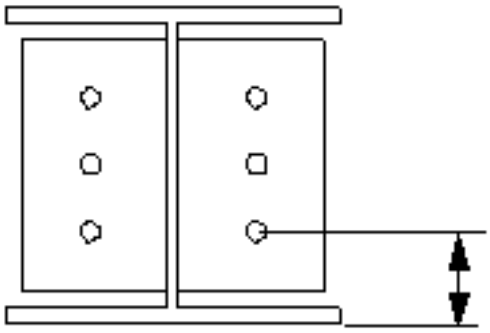
Use the **Bolts 1 - 2** tab to control the bolts that connect the end plates.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt. 

	Description
	• **Middle**: From the center line of the secondary part to the center line of the bolts.  • **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Dimension for vertical bolt group position.

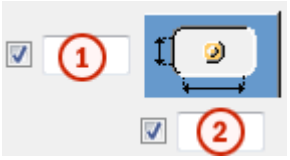
	Description
7	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



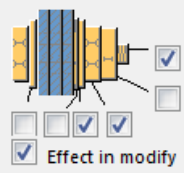
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	

Option	Description	Default
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

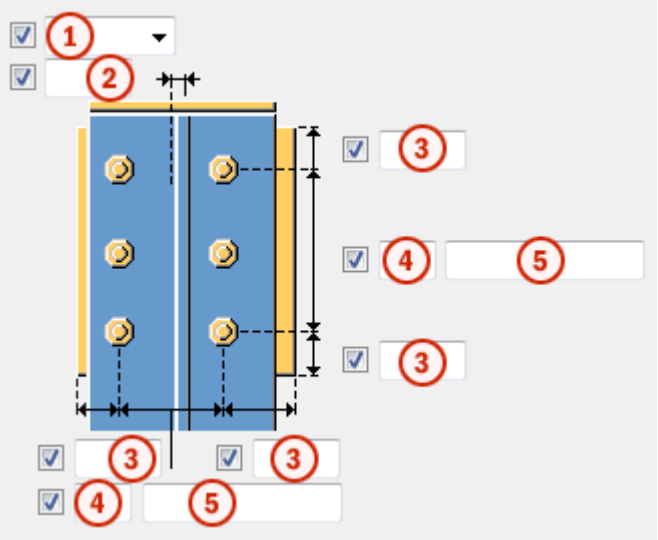
Bolt length increase

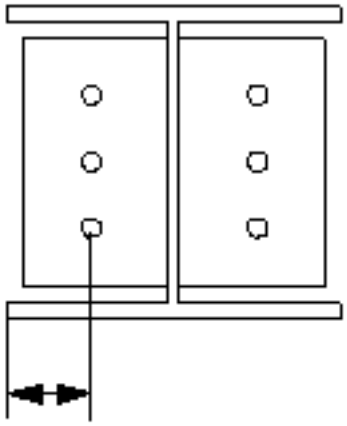
Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

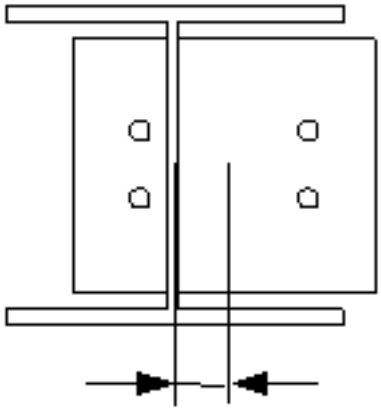
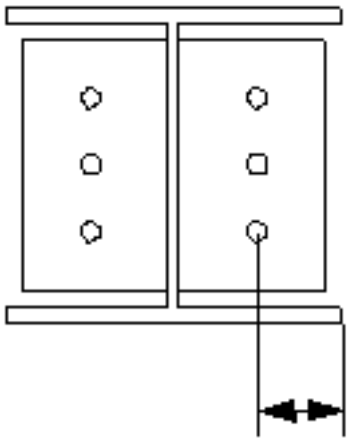
Bolts 3/Bolts 4 tabs

Use the **Bolts 3** and **Bolts 4** tabs to control the bolts that connect the first secondary beam or the second secondary beam to the column.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt. 

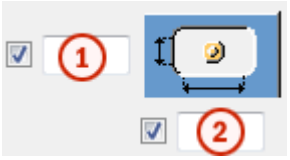
	Description
	• **Middle**: From the center line of the secondary part to the center line of the bolts.  • **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



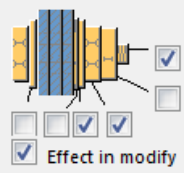
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	

Option	Description	Default
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Holes - Plate 1/Holes - Plate 2/Holes - Plates 3 and 4 tabs

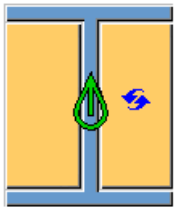
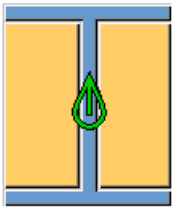
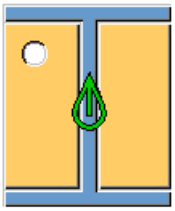
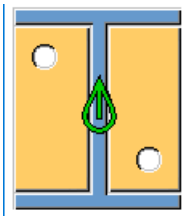
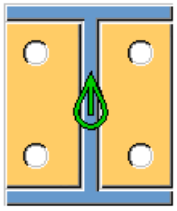
Use the **Holes** tabs to control the galvanizing holes in the end plates.

Option	Description
Bolt standard	Select the bolt standard.
Bolt type	Select the bolt type to define the location where the bolts should be attached.
Read data from	You can select to use the `sinkholes.dat` definition file to specify the default values for horizontal and vertical offsets, and the diameters for upper and lower holes. The file is searched in the following order: Environment common system steel folder (`..\Environments`

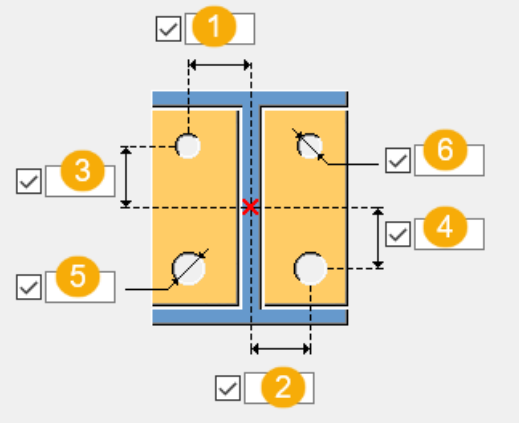
Option	Description
	\common\system\Steel), model folder, XS_FIRM, XS_PROJECT and XS_SYSTEM folder. You can also select to define the holes in the component dialog box.

Number of holes

The center of a hole group is the middle point of the beam and the middle point of the haunch, if the haunch exists. The hole groups are composed of 0, 1, 2 or 4 holes.

Option	Description
	Default No holes AutoDefaults can change this option.
	No holes
	1 hole
	2 holes
	4 holes

Hole positions



	Description
1	Plates 1 and 2: Horizontal distance between the secondary beam center and the upper hole. Plates 3 and 4: Horizontal distance between the main part center and the upper hole.
2	Plates 1 and 2: Horizontal distance between the secondary beam center and the lower hole. Plates 3 and 4: Horizontal distance between the main part center and the lower hole.
3	Plates 1 and 2: Vertical distance between the secondary beam center and the upper hole. Plates 3 and 4: Vertical distance between the main part center and the upper hole.
4	Plates 1 and 2: Vertical distance between the secondary beam center and the lower hole. Plates 3 and 4: Vertical distance between the main part center and the lower hole.
5	Diameter of the lower hole.
6	Diameter of the upper hole.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

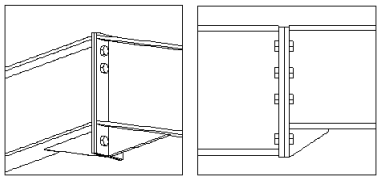
Joining plates (14)

Joining plates (14) connects two beam ends using an end plate welded to both beams. End plates are connected with bolts. Stiffeners and ear plates can also be created.

Objects created

- End plates
- Shim plates
- Ear plates
- Stiffeners
- Bolts
- Welds

Use for

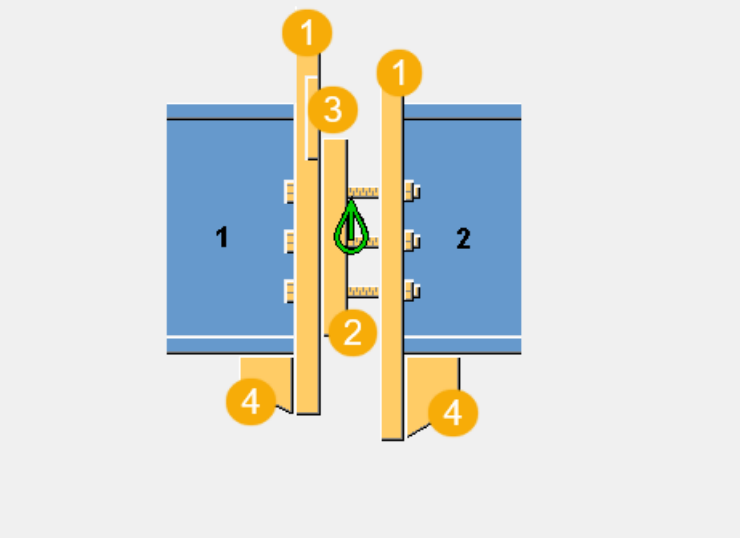
Situation	Description
	Beam ends connected with bolted end plates. Stiffener is created.

Selection order

1. Select the main part (beam).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

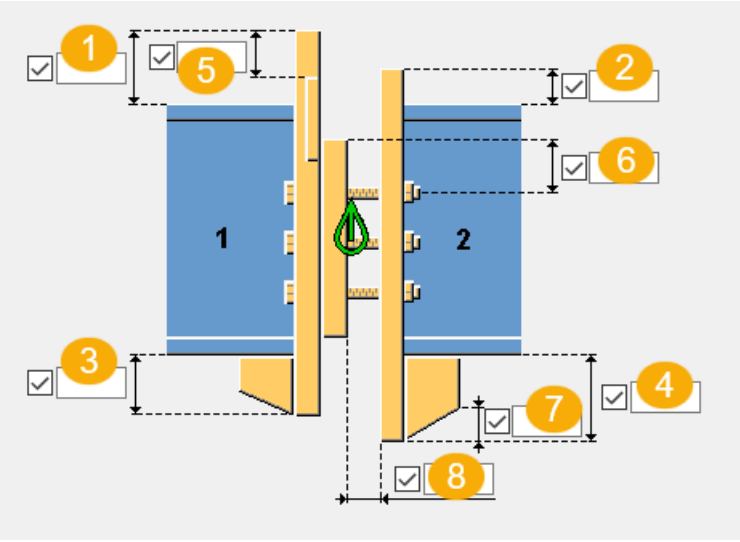


	Description
1	End plate
2	Shim plate
3	Ear plate
4	Stiffener

Picture tab

Use the **Picture** tab to define the geometry of the connection.

Dimensions



	Description	Default
1	End plate edge distance from the main part upper edge. If you do not enter a value for the end plate edge distance on the main part side, the secondary part side value is used on the main part side as well.	10 mm
2	End plate edge distance from the secondary part upper edge.	10 mm
3	End plate edge distance from the main part lower edge. If you do not enter a value for the end plate edge distance on the main part side, the secondary part side value is used on the main part side as well.	10 mm
4	End plate edge distance from the secondary part lower edge.	10 mm
5	Ear plate edge distance from the end plate upper edge.	38 mm
6	Shim plate vertical bolt edge distance.	1.5*bolt diameter
7	Stiffener edge inclination. To get a rectangular stiffener, set the value to 0. To get a triangular stiffener, enter the same value here as for the stiffener width on the **Parts** tab.	25 mm
8	Shim plate horizontal distance from the secondary part end plate.	

Parts tab

Use the **Parts** tab to define the part properties, and the shim plate shape and bolt edge distance.

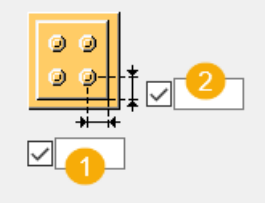
Parts

Option	Description
Sec end plate	Thickness, width, and height of the secondary part end plate.
Prim end plate	Thickness, width, and height of the main part end plate.
Fitting plate 1, **Fitting plate 2**, **Fitting plate 3**	Thickness of the shim plate.

Option	Description
Number of fitting pl. 1, **Number of fitting pl. 2**, **Number of fitting pl. 3**	Number of shim plates.
Ear plates	Thickness, width, and height of the ear plates. When you enter the ear plate thickness, two ear plates are by default created.
Stiffeners	Thickness and width of the stiffeners.

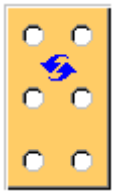
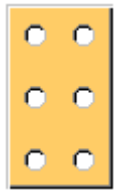
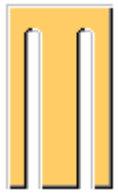
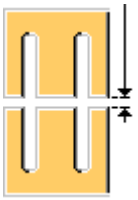
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Shim plate bolt edge distance



	Description
1	Horizontal bolt edge distance in the shim plate.
2	Vertical bolt edge distance in the shim plate.

Shim plate shape

Option	Description
	Default Holes are based on the bolt group of the connection. AutoDefaults can change this option.
	Holes are based on the bolt group of the connection.
	Finger shim plate with horizontal slots. The plate can be installed from the right or the left side of the connection.
	Finger shim plate with vertical slots. The plate can be installed from the top of the connection.
	Two separate finger shim plates with horizontal slots.
	Two separate finger shim plates with vertical slots.

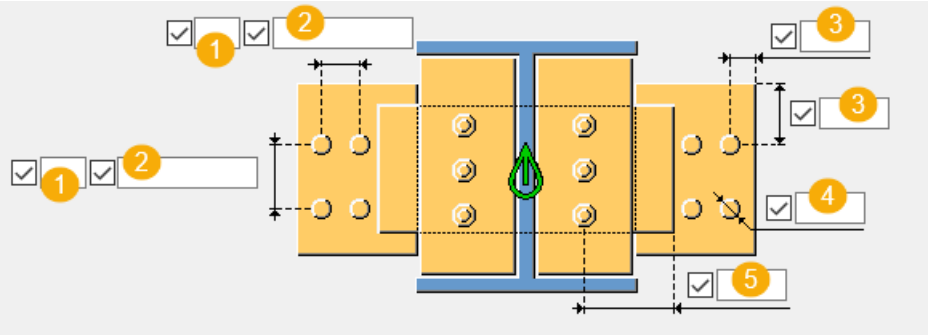
Tolerance

Define the tolerance for the slots in finger shim plates. The width of the slot is the bolt diameter + the tolerance. For two separate shim plates, also define the tolerance between the plates.

Parameters tab

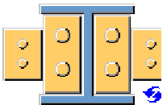
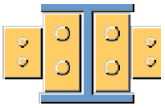
Use the **Parameters** tab to define ear plate creation and dimensions.

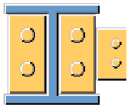
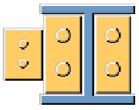
Dimensions



	Description	Default
1	Number of ear bolts You can define the number of the holes in the horizontal and vertical direction.	Horizontal: 1 Vertical: 2
2	Ear bolt spacing	3*hole diameter If there are more than two bolts in the bolt group in one direction, give the distances separated by a space (for example, 40 50).
3	Ear bolt edge distance Edge distance is calculated from the top and outside edges.	2.5*hole diameter
4	Ear bolt hole diameter	
5	Bolt edge distance to the edge of the shim plate	

Ear plate creation

Option	Description
	Default Both ear plates are created. AutoDefaults can change this option.
	Both ear plates are created.

Option	Description
	Ear plate 1 is created.
	Ear plate 2 is created.

Gap size

Define the limit value for the gap between the end plate and the secondary beam. Use this when the beam is slightly curved or sloped to decide if the end angle is so small that the beam end can be straight.

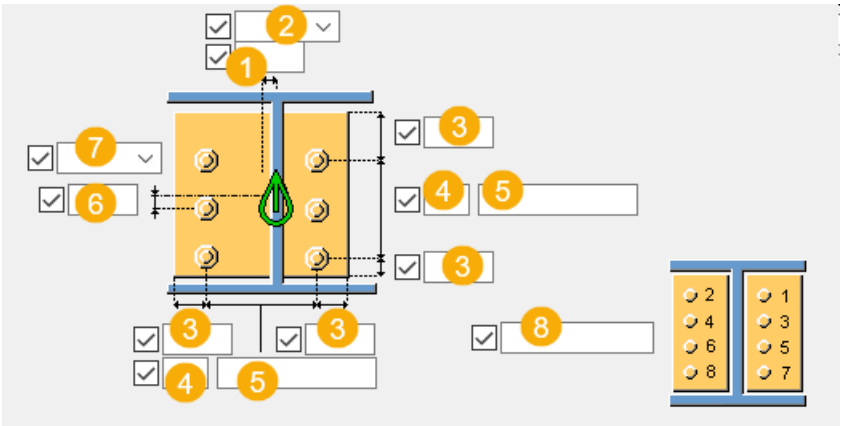
If the actual gap is smaller than this value, the end of the beam is left straight.

If the actual gap is larger than this value, the end of the beam is fitted to the end plate.

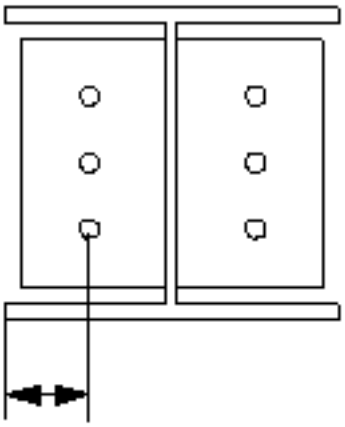
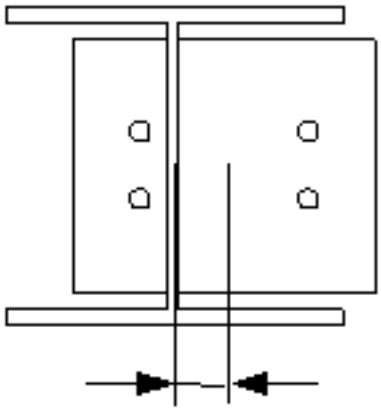
Bolts tab

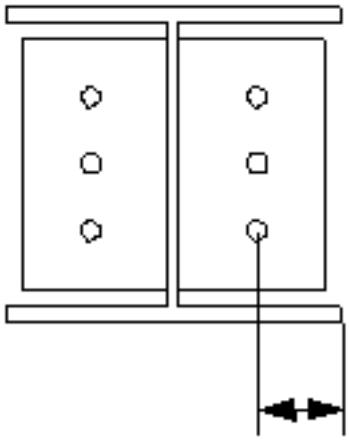
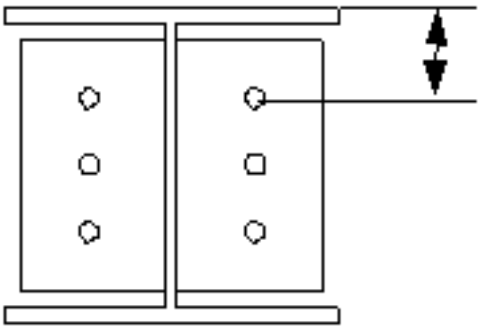
Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

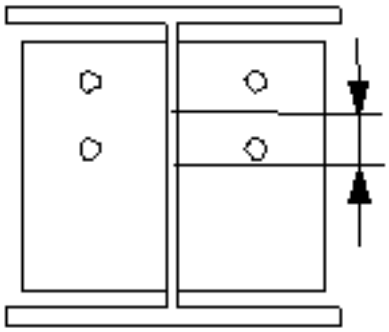
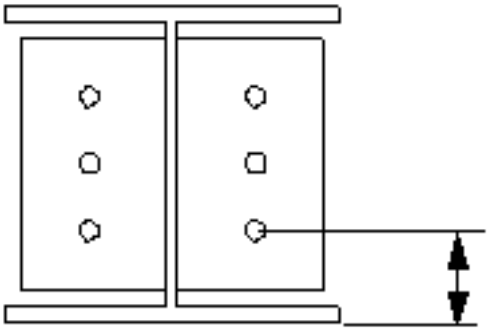
Bolt group dimensions



1	Dimension for horizontal bolt group position.

2	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt.  • **Middle**: From the center line of the secondary part to the center line of the bolts. 

	• **Right**: From the right edge of the secondary part to the rightmost bolt. 
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Dimension for vertical bolt group position.
7	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
8	Define which bolts are deleted from the bolt group.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

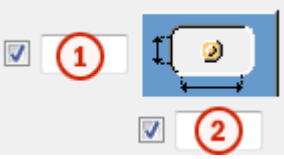
Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Bolt comment

You can define a bolt comment.

Slotted holes

You can define slotted, oversized, or tapped holes.

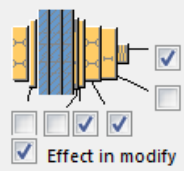


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Holes tab

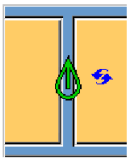
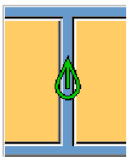
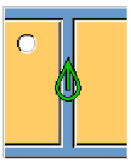
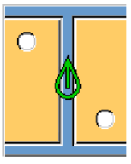
Use the **Holes** tab to control the galvanizing holes in the end plates.

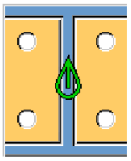
Option	Description
Bolt standard	Select the bolt standard.
Bolt type	Select the bolt type to define the location where the bolts should be attached.

Option	Description
Read data from	You can select to use the `sinkholes.dat` definition file to specify the default values for horizontal and vertical offsets, and the diameters for upper and lower holes. The file is searched in the following order: Environment common system steel folder (`..\Environments\common\system\Steel`), model folder, `XS_FIRM`, `XS_PROJECT` and `XS_SYSTEM` folder. You can also select to define the holes in the component dialog box.

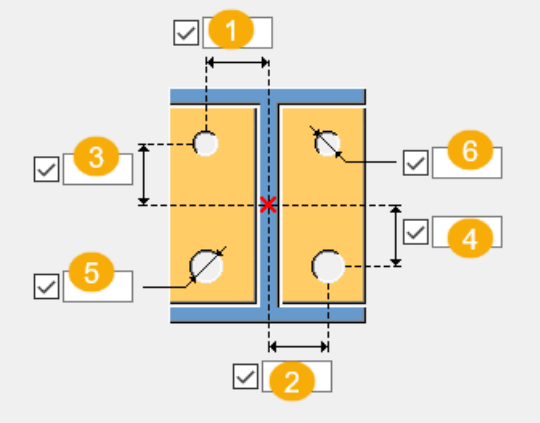
Number of holes

The center of a hole group is the middle point of the beam and the middle point of the haunch, if the haunch exists. The hole groups are composed of 0, 1, 2, or 4 holes.

Option	Description
	Default No holes AutoDefaults can change this option.
	No holes
	1 hole
	2 holes

Option	Description
	4 holes

Hole positions



1	Horizontal distance between the end plate center and the upper hole.
2	Horizontal distance between the end plate center and the lower hole.
3	Vertical distance between the end plate center and the upper hole.
4	Vertical distance between the end plate center and the lower hole.
5	Diameter of the lower hole.
6	Diameter of the upper hole.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Dstv connection properties

Click the link below to find out more:

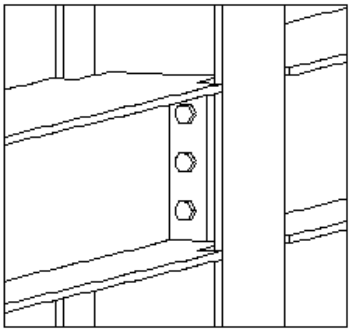
Two-sided end plate (24)

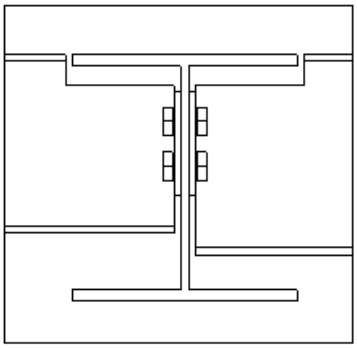
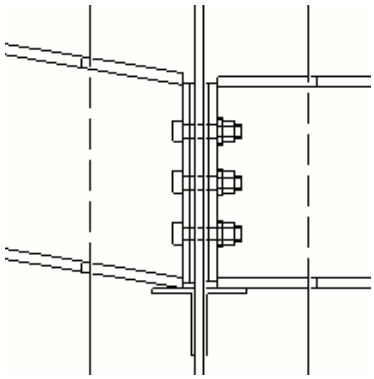
Two-sided end plate (24) connects two beams to a column or to a beam using a bolted end plate. One bolt group goes through all the three parts. This connection is created and used in the same way as **End plate (29)** connection.

Objects created

- End plate
- Seat (plate or angle)
- Shim plates
- Stiffeners
- Welds
- Bolts
- Cuts

Use for

Situation	Description
	Beam-to-column end plate connection.

Situation	Description
	Beam-to-beam end plate connection.
	End plate connection with shim plates and a seat angle.

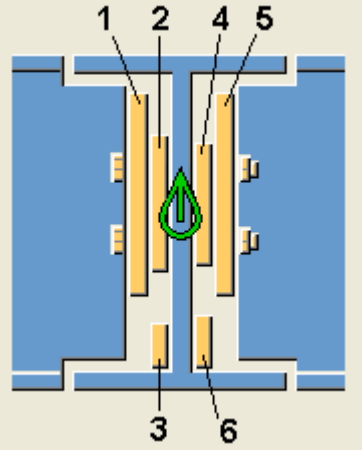
Limitations

Do not use this connection when connecting a beam to a column flange.

Selection order

1. Select the main part (column or beam).
2. Select the first secondary part (beam).
3. Select the second secondary part (beam).
4. Click the middle mouse button to create the connection.

Part identification key

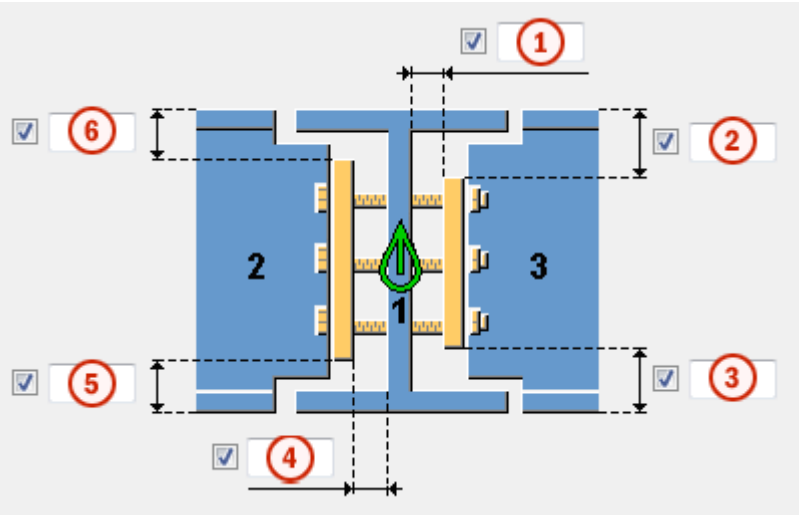


	Part
1	End plate for the first secondary beam
2	Shim plate for the first secondary beam
3	Seat (plate or angle) for the first secondary beam
4	Shim plate for the second secondary beam
5	End plate for the second secondary beam
6	Seat (plate or angle) for the second secondary beam

Picture tab

Use the **Picture** tab to control the position of the end plate.

End plate position



	Description
1	Gap between the main part and the end plate or the shim plate of the second secondary beam.
2 **3**	End plate edge distance from the top or bottom of the second secondary beam.
4	Gap between the main part and the end plate or the shim plate of the first secondary beam.
5 **6**	End plate edge distance from the top or the bottom of the first secondary beam.

Parts tab

Use the **Parts** tab to define the properties of the parts created.

Plate

Option	Description	Default
End plate	End plate thickness, width and height. The width and height are defined by the bolt group edge distances. The height can also be entered as a plate edge distance from the upper and lower edge of the secondary beam.	`half of the bolt diameter`
Seat	Define whether a seat is created under the end plate. The seat is created only if the seat thickness is entered. Adding a seat plate moves the end plate by default 20 mm below the secondary beam bottom.	width = `end plate width` height = `secondary beam height`
L seat	Define whether a seat angle is created under the end plate. The length	width = `end plate width`

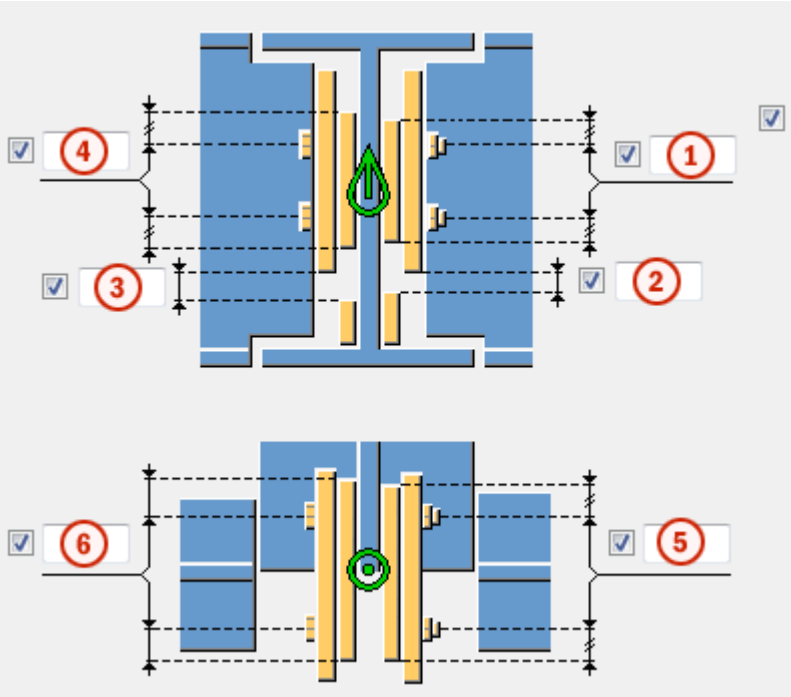
Option	Description	Default
	of the angle is defined by the seat width. If you enter a seat angle profile, the seat will be an angle instead of a plate even if you have entered the plate thickness. Adding a seat angle does not move the end plate in the same way as the seat plate. To avoid collision between the angle and the secondary beam, modify the end plate or enter a gap between the end plate and the seat angle.	
Fitting plate	Shim plate thickness. The plate is created only if the plate thickness is entered.	width and height = `defined by the bolt group and the shim plate edge distances`
Number or fitting pl.	Define how many shim plates are created.	

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

Use the **Parameters** tab to control the shim plate and end plate positions.

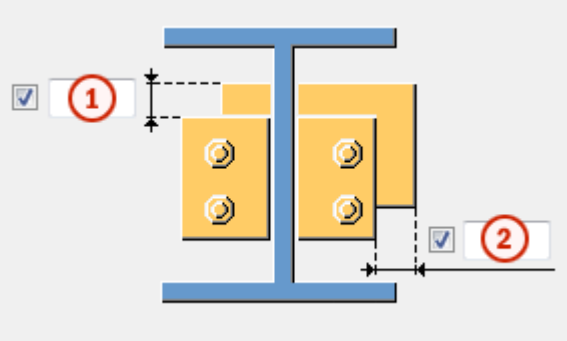
End plate dimensions



	Description	Default
1 **4**	Vertical bolt edge distance for the shim plates. The number and spacing of bolts affect the size of the shim plates.	`1.5*bolt diameter`
2 **3**	Gap between the end plate and the seat (plate or angle).	`0 mm`
5 **6**	Horizontal bolt edge distance for the shim plates.	

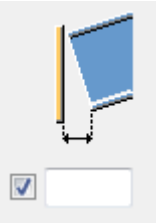
Shim plate and end plate positions

Define the position of the shim plates and the end plate in the second secondary beam. The plates move in relation to the plates in the first secondary beam. By default, the plates in the second secondary beam are positioned so that holes are placed symmetrically. You may need to move the plates, for example, when connecting skew or curved secondary beams.



	Description
1	Define how much the plates are moved in vertical direction.
2	Define how much the plates are moved in horizontal direction.

Gap size



Define the limit value for the gap between the end plate and the secondary or main beam. Use this gap when the beam is slightly curved or sloped to decide if the end angle is so small that the beam end can be straight.

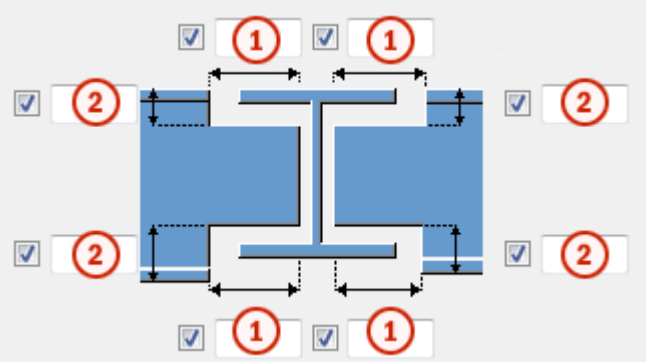
If the actual gap is smaller than this value, the beam end is left straight.

If the actual gap is larger than this value, the beam end is fitted to the end plate.

Notch tab

Use the **Notch** tab to modify the horizontal and vertical cuts.

Cut dimensions

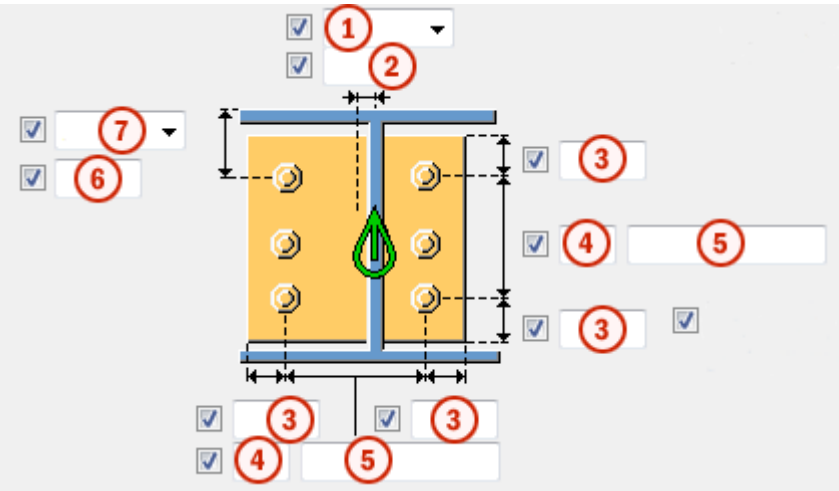


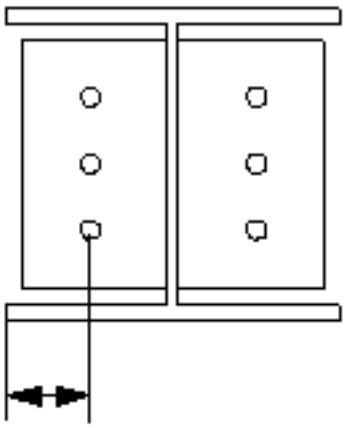
	Description	Default
1	Dimensions for the horizontal flange cuts.	10 mm
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest 5 mm.

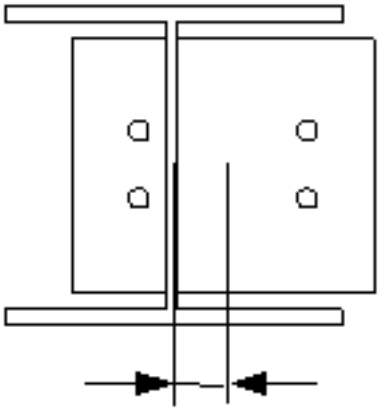
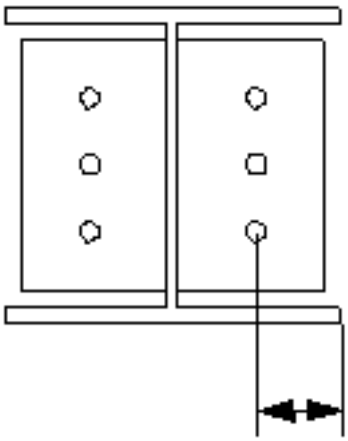
Bolts tab

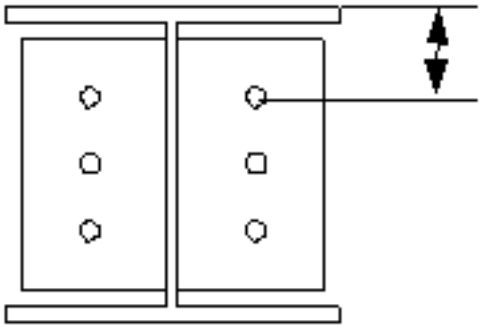
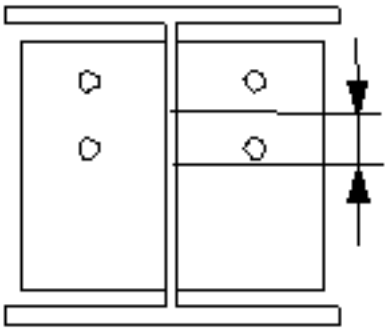
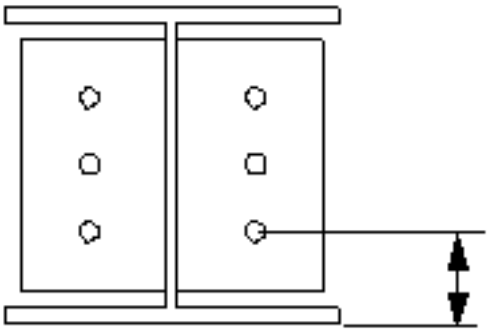
Use the **Bolts** tab to control the properties of the bolts used.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt. 

	Description
	• **Middle**: From the center line of the secondary part to the center line of the bolts.  • **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Dimension for vertical bolt group position.

	Description
7	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

Staggering of bolts

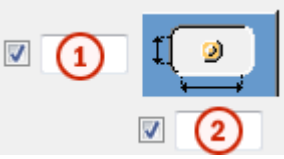
Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

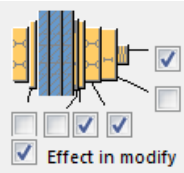


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

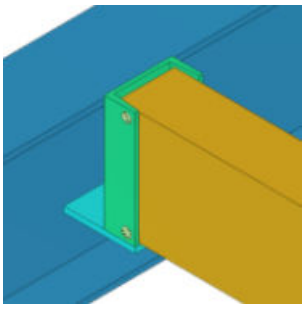
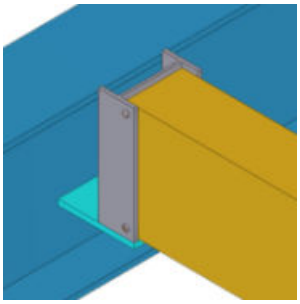
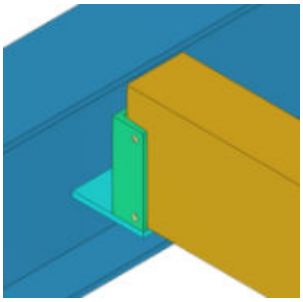
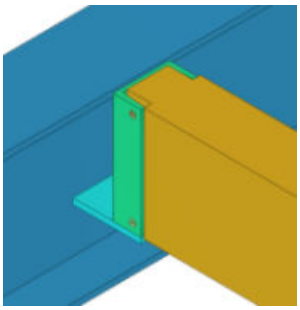
Wooden purlin shoe (15)

Wooden purlin shoe (15) creates a connection between a steel beam and a wooden purlin. A purlin shoe is welded to the steel beam, in which a wooden purlin is fitted. You can use various profiles to create the purlin shoe, for example, U-shaped and L-shaped profiles. The purlin shoe has holes so the wooden purlin can be connected using hexagon head wood screws.

Objects created

- Purlin shoe
- Bolts
- Welds

Use for

Situation	Description
	Wooden purlin shoe, U profile.
	Wooden purlin shoe, I profile.
	Wooden purlin shoe.
	Wooden purlin shoe, purlin is notched.

Before you start

Create a steel beam and a purlin.

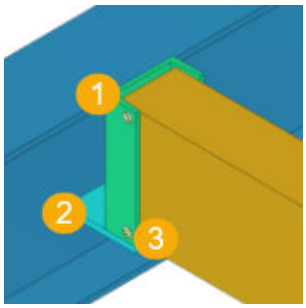
Selection order

1. Select the beam (main part).

2. Select the wooden purlin (secondary part).

   The connection is created automatically when the secondary part is selected.

Part identification key

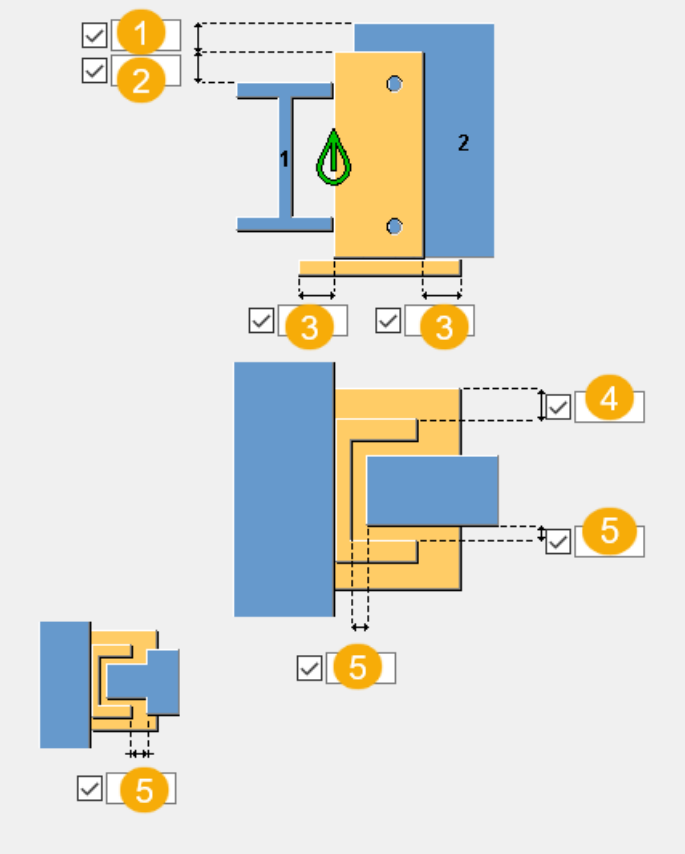


	Description
1	Purlin shoe profile
2	Bottom plate
3	Bolt group

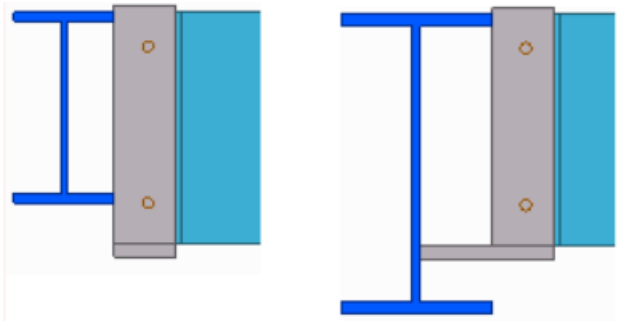
Picture tab

Use the **Picture** tab to define the offset dimensions of the shoe profile, base plate, and notch.

Offset dimensions



	Description	Default
1	Shoe profile offset from the purlin top.	
2	Shoe profile offset from the steel beam top. This dimension takes priority over the shoe profile offset.	5 mm
3	Bottom plate offset in the direction of the wooden purlin. If you do not enter a value and the steel beam bottom is further down than the wooden purlin bottom, the bottom plate extends automatically to the main part web. For example:	0

	Description	Default
		
4	Bottom plate offset in the direction of the main part.	0
5	Notch offset. When the shoe profile width is insufficient, the wooden purlin is notched.	

Parts tab

Use the **Parts** tab to define the shoe profile and bottom plate properties.

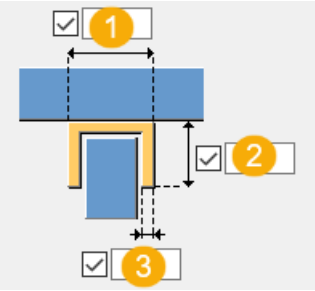
Parts

Option	Description
Bottom plate	Thickness of the bottom plate.
Profile	Select the shoe section profile from the profile catalog. If you do not select any profile, the shoe profile shape selected in **Section type** is used.
Section type	Select the shoe profile section type. Section type is used when a profile is not defined in **Profile**. Which profile is used depends on the wooden purlin width. The shoe profile has to be chosen so that the wooden purlin fits the shoe without needing to notch the wooden purlin. Catalog profiles are used, except for the last two options: U-shaped bent plate and L-shaped bent plate. When you use either of these, you can select in **Create shoe as** whether the U-

Option	Description
	shaped or L-shaped profile is created as a folded plate or a polybeam. If you create a polybeam, you can create an unfolded bent plate drawing with bent lines.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Section dimensions



	Description
1	Shoe profile width. If you have selected a catalog section (`UNP`, `UPE`, `INP`, `IPE`, `L`) in **Section type** you have to define a width that corresponds with the catalog profiles.

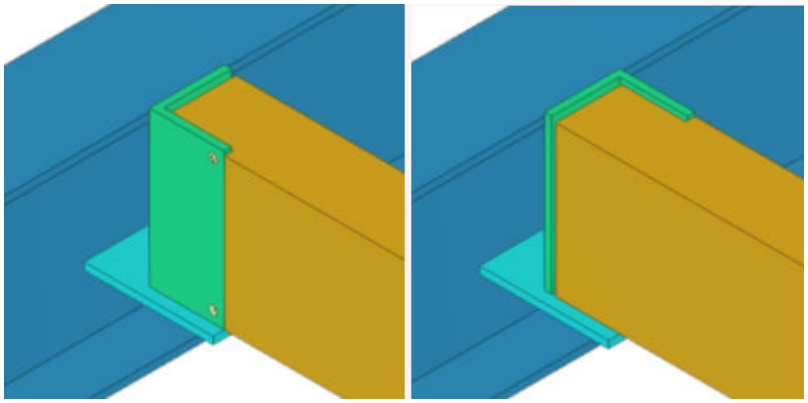
	Description
	For example, if you have selected the `UNP` section, you can set the width to `80` because this profile exists in the catalog. You cannot use value `90` because the catalog does not contain a `UNP90` profile. If you use a value that is not valid, the shoe profile will disappear.
2	Shoe profile flange height. You can define the flange height for U-shaped bent plates and L-shaped bent plates.
3	Shoe profile flange thickness. You can define the flange thickness for U-shaped bent plates and L-shaped bent plates.

Shoe reference line

Define the position of the shoe profile reference line.

L position

Select the shoe profile position if the profile is L shaped. The options are left or right:



Create shoe as

If you have selected a U-shaped bent plate or an L-shaped bent plate in **Section type**, you can select whether the plate is created as a folded plate or a polybeam.

NOTE Select **Polybeam** to create an unfolded bent plate drawing with bent lines.

If you select **Folded plate**, you cannot create an unfolded bent plate drawing with bent lines.

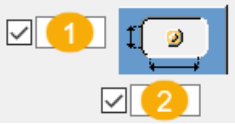
Holes tab

Use the **Holes** tab to define the hole properties and the pattern in the shoe profile.

Basic properties

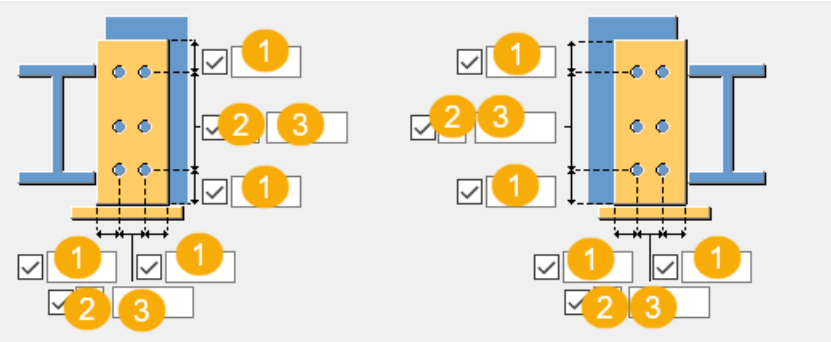
Option	Description	Default
Bolt standard	Bolt assembly standard to be used inside the component. The available standards are defined in the bolt assembly catalog.	The default value is `4014-8.8`.
Bolt size	Hole diameter.	`18 mm`
Assembly type	Location where the bolts should be attached.	**Site**

Slotted holes



	Description	Default
1	Vertical dimension of the slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of the slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Dimensions

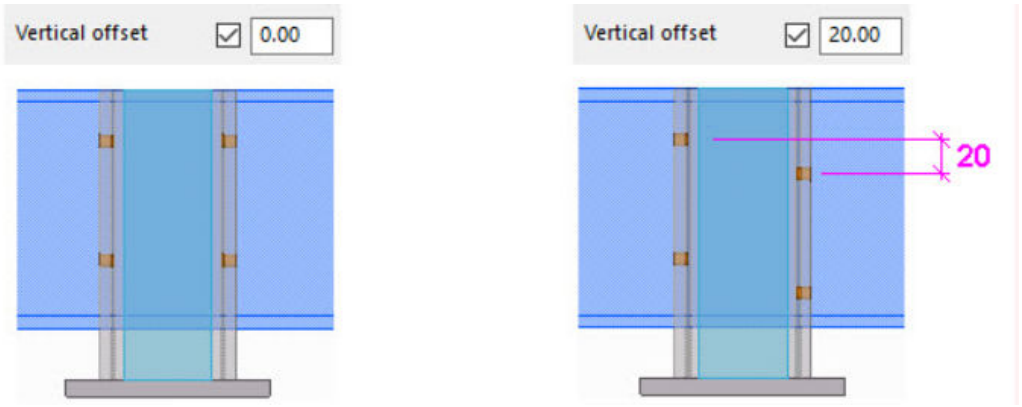


	Description
1	Hole edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of holes.
3	Hole spacing. Use a space to separate the spacing values. Enter a value for each space between holes. For example, if there are 3 holes, enter 2 values.

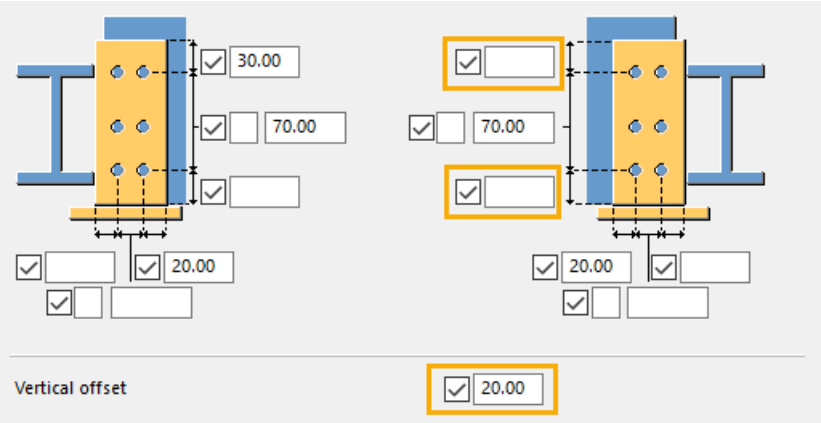
Vertical offset

Define the vertical shift between the bolt groups.

If the purlin is screwed at the same height on both sides of the shoe profile, the screws may hit each other. To prevent this, one hole pattern can be vertically shifted. In this way, the screws cannot hit each other. The default value is `0`. For example:



Note that the vertical offset can only be used when you do not enter a vertical edge distance for the bolt group to be shifted. If you enter a value for these edge distances, the edge distance is applied and the hole pattern is not shifted vertically.



Extra weld parameters tab

Use the **Extra weld parameters** tab to define how the shoe profile is connected to the main part.

Connect parts as

NOTE Set the desired connection type before adding the component in the model.

If you have to change the connection type after you have added the component in the model, you have to delete the component and recreate it with the changed connection type.

Option	Description
Secondary part of welding	The shoe profile is dimensioned and labeled as a welded part in the assembly drawing. It is not possible to create a separate assembly drawing of the shoe profile.
Sub-assembly	The shoe profile is added as a sub-assembly. It is possible to create a separate assembly drawing of the shoe profile. In the assembly drawing of the main part, the position of the sub-assembly is dimensioned but the dimensions of the sub-assembly parts are not. This is because this sub-assembly is already fabricated when it is welded to the main part. This results in better readable assembly drawings.

General tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Welds

Click the link below to find out more:

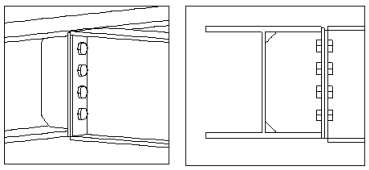
Stiffened end plate (27)

Stiffened end plate (27) connects two beams, or a beam to a column, using an end plate at the secondary beam end and a T-shaped plate construction consisting of a stiffener and a shear tab. The plates are connected with bolts.

Objects created

- End plate
- Shear tabs
- Stiffeners
- Shim plates
- Bolts
- Welds

Use for

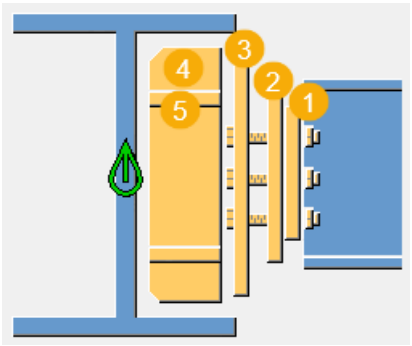
Situation	Description
	Two beams connected with a bolted end plate. Shear tab is created.

Selection order

1. Select the main part (beam or column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key



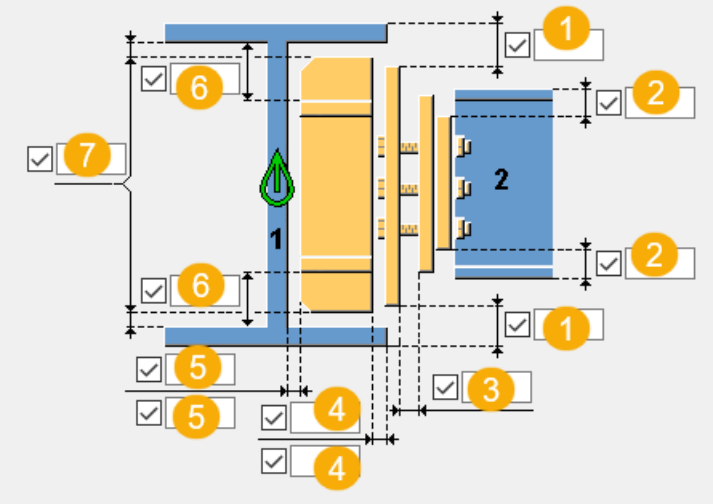
	Description
1	End plate

	Description
2	Shim plate
3	Front plate
4	Shear tab
5	Stiffener

Picture tab

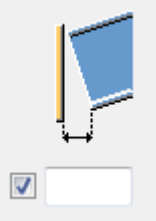
Use the **Picture** tab to define the part positions.

Dimensions



	Description
1	Front plate edge distance from the main part flange.
2	End plate edge distance from the secondary part flange.
3	Gap between the front plate and the shim plate.
4	Shear tab edge distance from the main part flange. This dimension has an effect on the shear tab size. The lower box is for controlling the shear tab on the other side of the main part.
5	Shear tab edge distance from the main part web. The lower box is for controlling the shear tab on the other side of the main part.
6	Stiffener edge distance from the shear tab edge.
7	Shear tab edge distance from the main part flange. This dimension has an effect on the shear tab size.

Gap size



Define the limit value for the gap between the end plate and the secondary beam. Use this when the beam is slightly curved or sloped to decide if the end angle is so small that the beam end can be straight.

If the actual gap is smaller than this value, the end of the beam is left straight.

If the actual gap is larger than this value, the end of the beam is fitted to the end plate.

Parts tab

Use the **Parts** tab to define the part properties.

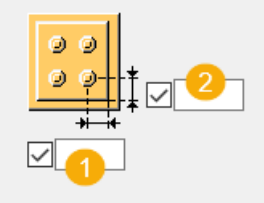
Parts

Option	Description	Default
End plate	Thickness of the end plate.	`10 mm`
Front plate	Thickness and width of the front plate.	Thickness: `10 mm` Width: Same as the width of the end plate
Shear plate	Thickness, width and height of the shear tab.	
Fill shear plate width	Select whether to use the `XS_STANDARD_STIFFENER_WIDTH_TOLERANCE` advanced option for the width.	
Stiffener	Thickness and width of the stiffener.	
Bottom stiffener as one plate	Select whether the bottom stiffener is created as one plate.	
Fitting plate 1, **Fitting plate 2**, **Fitting plate 3**	Thickness of the shim plate.	
Number of fitting pl. 1, **Number of fitting pl. 2**,	Define the number of shim plates.	

Option	Description	Default
Number of fitting pl. 3		

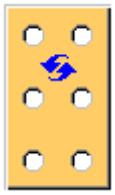
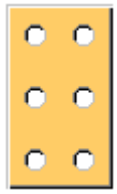
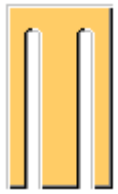
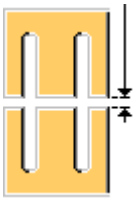
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Shim plate bolt edge distance



	Description
1	Horizontal bolt edge distance in the shim plate.
2	Vertical bolt edge distance in the shim plate.

Shim plate shape

Option	Description
	Default Holes are based on the bolt group of the connection. AutoDefaults can change this option.
	Holes are based on the bolt group of the connection.
	Finger shim plate with horizontal slots. The plate can be installed from the right or the left side of the connection.
	Finger shim plate with vertical slots. The plate can be installed from the top of the connection.
	Two separate finger shim plates with horizontal slots.
	Two separate finger shim plates with vertical slots.

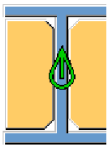
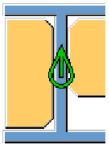
Tolerance

Define the tolerance for the slots in finger shim plates. The width of the slot is the bolt diameter + the tolerance. For two separate shim plates, also define the tolerance between the plates.

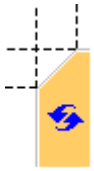
Parameters tab

Use the **Parameters** tab to define the chamfers of the shear tab and stiffeners.

Shear tab shape

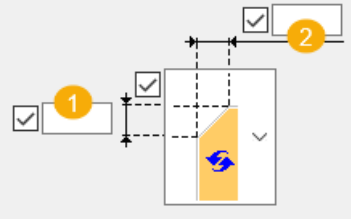
	Default Full Creates a full shear tab of the same height as the web of the main part. AutoDefaults can change this option.
	Full
	Shear tabs are created on both sides of the main part web.
	Partial shear tab is created on the other side of the main part web.

Chamfer type

Option	Description
	Default. Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer

Option	Description
	Convex arc chamfer
	Concave arc chamfer

Chamfer dimensions

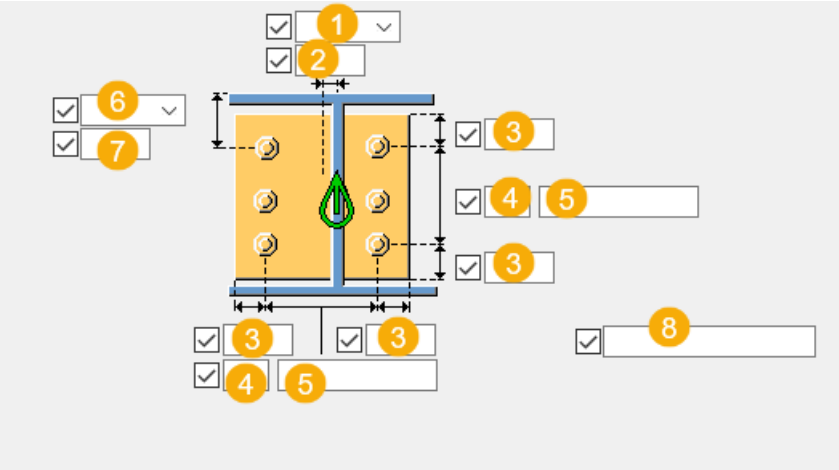


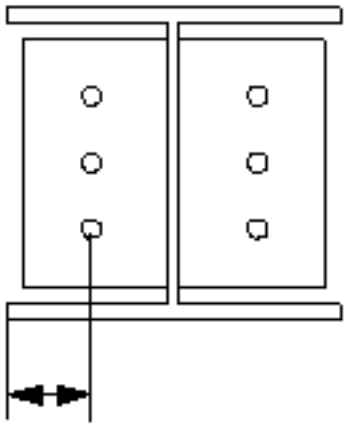
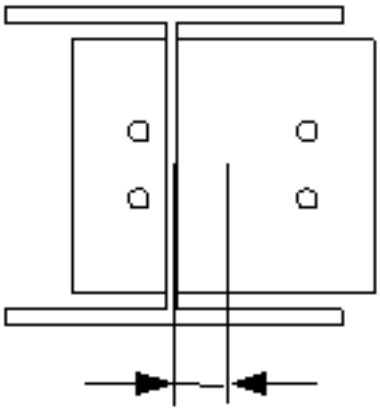
	Description	Default
1	Vertical dimension of the chamfer.	`10 mm`
2	Horizontal dimension of the chamfer.	`10 mm`

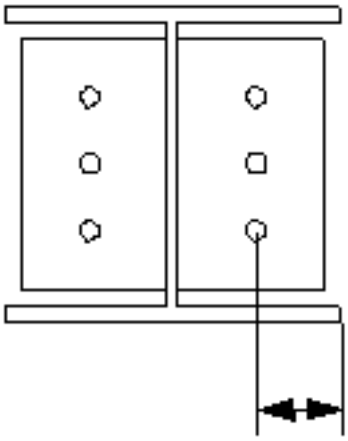
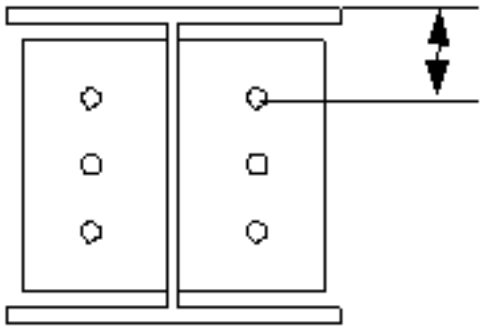
Bolts tab

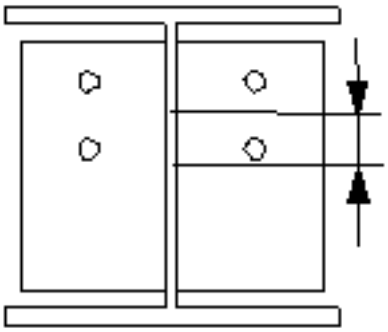
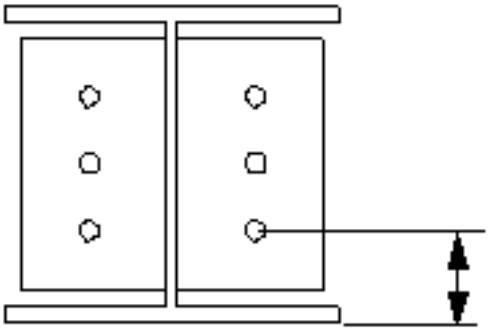
Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt.  • **Middle**: From the center line of the secondary part to the center line of the bolts. 

	Description
	• **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Dimension for vertical bolt group position.
8	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	

Option	Description	Default
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Bolt type

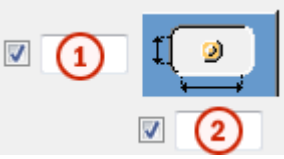
Select the bolt type to define the location where the bolts should be attached.

Bolt comment

You can define a bolt comment.

Slotted holes

You can define slotted, oversized, or tapped holes.

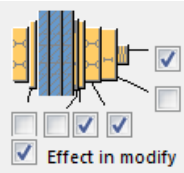


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Bolting direction

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1
	Bolting direction 2

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered

Option	Description
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

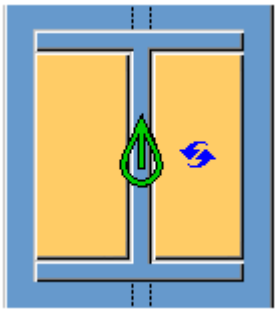
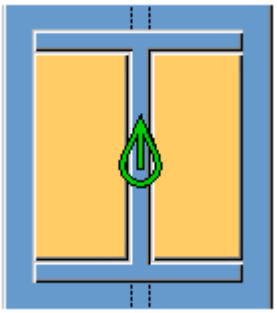
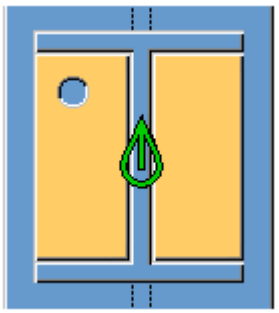
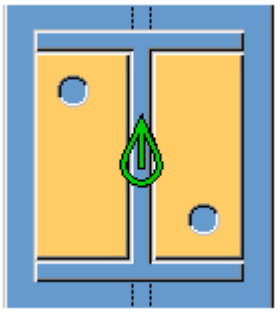
Holes - end plate tab

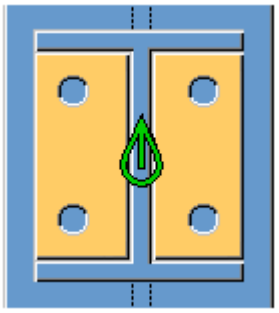
Use the **Holes - end plate** tab to control the galvanizing holes in the end plate.

Option	Description
Bolt standard	Select the bolt standard.
Bolt type	Select the bolt type to define the location where the bolts should be attached.
Read data from	You can select to use the `sinkholes.dat` definition file to specify the default values for horizontal and vertical offsets, and the diameters for upper and lower holes. The file is searched in the following order: Environment common system steel folder (`..\Environments\common\system\Steel`), model folder, `XS_FIRM`, `XS_PROJECT` and `XS_SYSTEM` folder. You can also select to define the holes in the component dialog box.

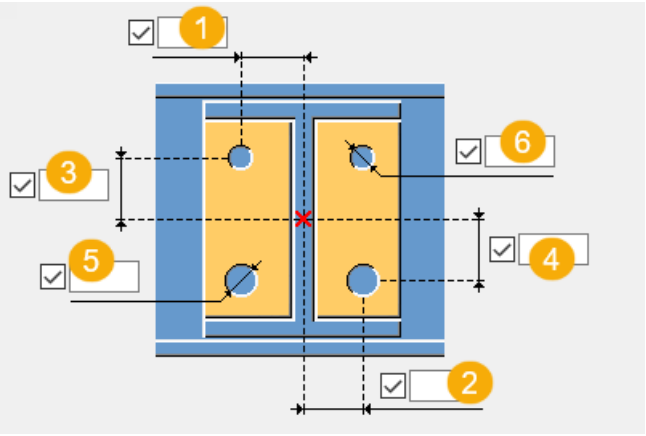
Number of holes

The center of a hole group is the middle point of the beam and the middle point of the haunch, if the haunch exists. The hole groups are composed of 0, 1, 2 or 4 holes.

Option	Description
	Default No holes AutoDefaults can change this option.
	No holes
	1 hole
	2 holes

Option	Description
	4 holes

Hole positions



	Description
1	Horizontal distance between the secondary beam center and the upper hole.
2	Horizontal distance between the secondary beam center and the lower hole.
3	Vertical distance between the secondary beam center and the upper hole.
4	Vertical distance between the secondary beam center and the lower hole.
5	Diameter of the lower hole.
6	Diameter of the upper hole.

Holes - front plate tab

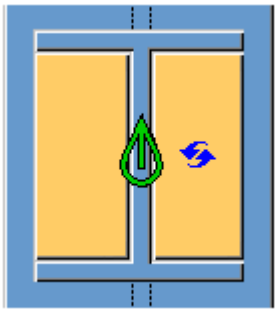
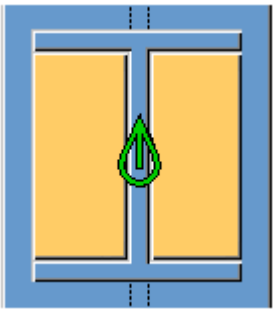
Use the **Holes - front plate** tab to control the galvanizing holes in the front plate.

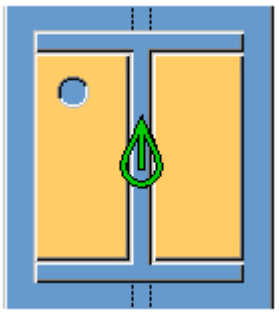
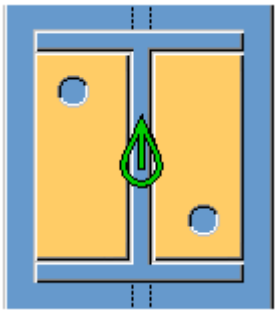
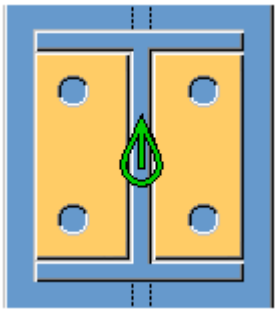
Option	Description
Bolt standard	Select the bolt standard.

Option	Description
Bolt type	Select the bolt type to define the location where the bolts should be attached.
Read data from	You can select to use the `sinkholes.dat` definition file to specify the default values for horizontal and vertical offsets, and the diameters for upper and lower holes. The file is searched in the following order: Environment common system steel folder (`..\Environments\common\system\Steel`), model folder, `XS_FIRM`, `XS_PROJECT` and `XS_SYSTEM` folder. You can also select to define the holes in the component dialog box.

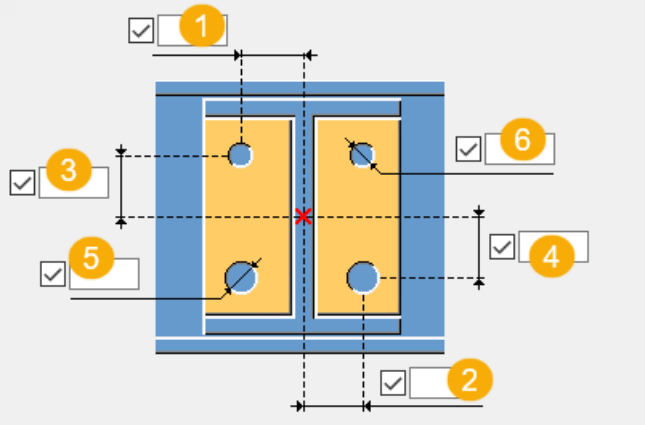
Number of holes

The center of a hole group is the middle point of the beam and the middle point of the haunch, if the haunch exists. The hole groups are composed of 0, 1, 2 or 4 holes.

Option	Description
	Default No holes AutoDefaults can change this option.
	No holes

Option	Description
	1 hole
	2 holes
	4 holes

Hole positions



	Description
1	Horizontal distance between the main beam center and the upper hole.
2	Horizontal distance between the main beam center and the lower hole.
3	Vertical distance between the main beam center and the upper hole.
4	Vertical distance between the main beam center and the lower hole.
5	Diameter of the lower hole.
6	Diameter of the upper hole.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Dstv connection properties

Click the link below to find out more:

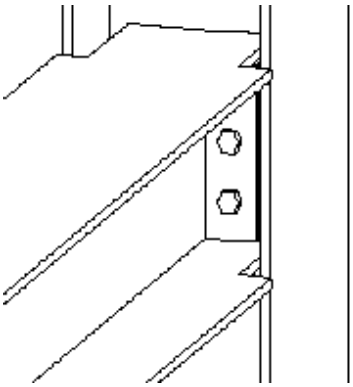
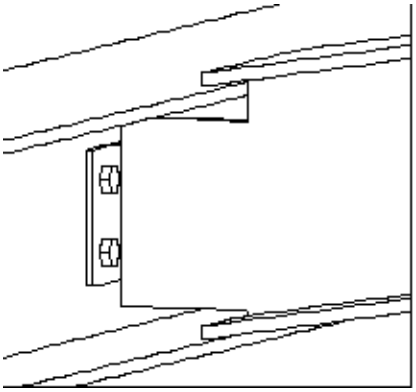
End plate (29)

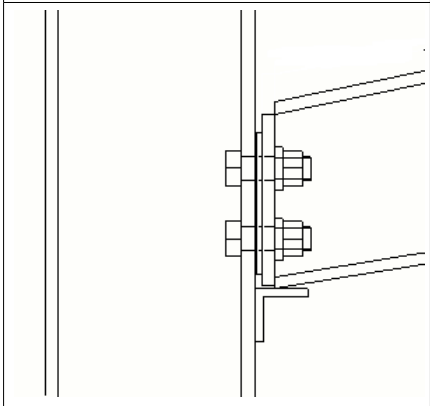
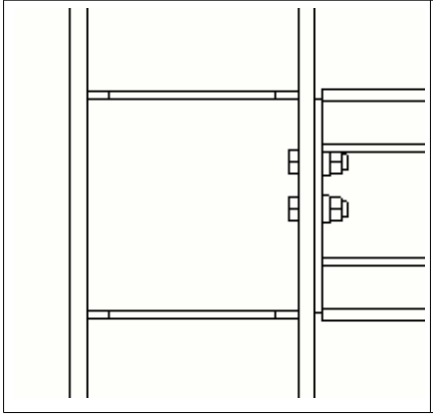
End plate (29) connects a beam to a column or two beams to each other using a bolted end plate. Stiffeners, seats and shim plates are optional.

Objects created

- End plate
- Seat angle or seat plate (optional)
- Shim plates (optional)
- Stiffeners (optional)
- Bent plate (optional)
- Welds
- Bolts
- Cuts

Use for

Situation	Description
	Beam-to-column end plate connection.
	Beam-to-beam end plate connection.

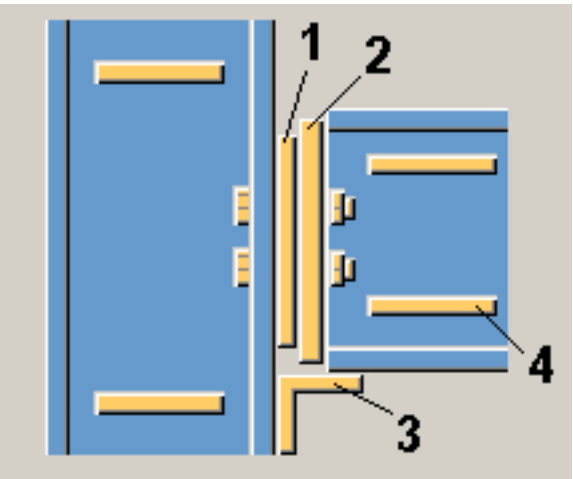
Situation	Description
	End plate connection with a shim plate and a seat angle.
	End plate connection with stiffeners.

Selection order

1. Select the main part (column or beam).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary beam is selected.

Part identification key

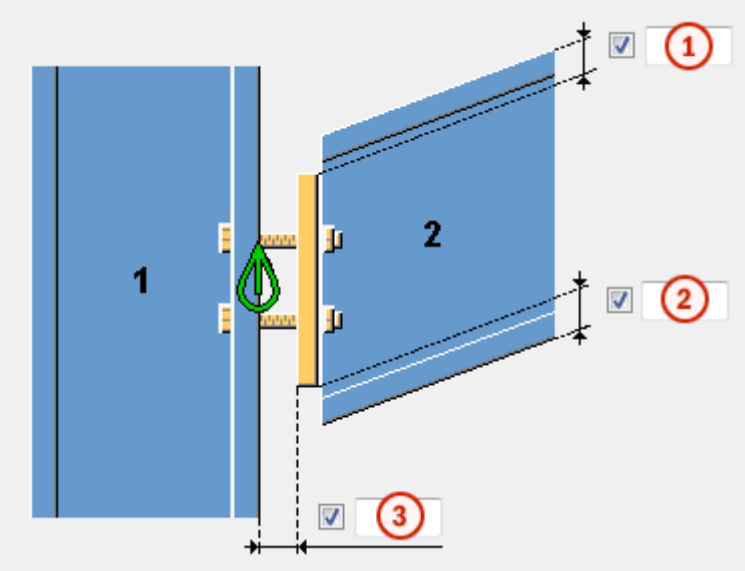


	Part
1	Shim plate
2	End plate
3	Seat (plate or angle)
4	Stiffener

Picture tab

Use the **Picture** tab to control the position of the end plate.

End plate position

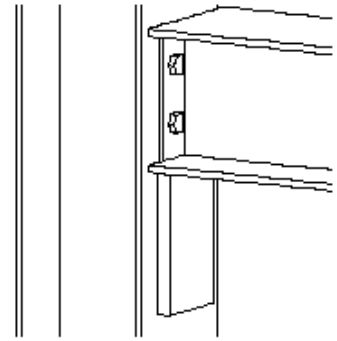


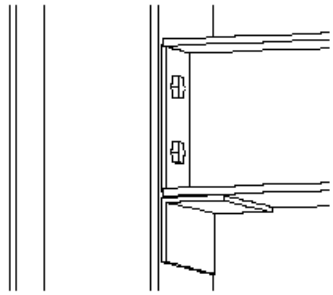
	Description	Default
1	End plate top position from the top of the secondary beam. If no value is entered, bolts and bolt edge distances define the size of the end plate. If you give both values, the position distance override the bolt edge distance values.	10 mm
2	End plate bottom position from the bottom of the secondary beam.	
3	Gap between the shim plate and the main part. If the shim plate does not exist, the defined gap is created between the end plate and the main part.	0 mm

Parts tab

Use the **Parts** tab to define the properties of the parts.

Plate

Part	Description	Default
End plate	End plate thickness, width and height. The width and height are defined by the bolt group edge distances. The height can also be entered as a plate edge distance from the upper and lower edge of the secondary beam.	`half of the bolt diameter`
Seat	Define whether a seat is created under the end plate. The seat is created only if the seat thickness is entered.  Adding a seat plate moves the end plate by default 20 mm below the secondary beam bottom.	width = `end plate width` height = `secondary beam height`
L seat	Define whether a seat angle is created under the end plate. The length	width = `end plate width`

Part	Description	Default
	of the angle is defined by the seat width.  If you enter a seat angle profile, the seat will be an angle instead of a plate even if you have entered the plate thickness. Adding a seat angle does not drag the end plate in the same way as the seat plate. To avoid collision between the angle and the secondary beam, modify the end plate or enter a gap between the end plate and the seat angle.	
Fitting Plate	Shim plate thickness. The plate is created only if the plate thickness is entered.	width = `defined by the bolt group and the shim plate edge distances`
Number of fitting pl. (DEF=1)	Define how many shim plates are created.	
Folded Plate	Bent plate thickness, width and height. The plate is created only if the plate thickness is entered. Bent plate can also be used with ordinary shim plates.	width = `determined from the bolt group inner bolts`

Part	Description	Default
Stiffeners	Define whether stiffeners are created to the column when a beam is connected to the column flange. If you enter one of the dimensions, stiffeners are created.	thickness = `20 mm` width = `width of the column flange` height = `height of the column web`
Upper horiz stiff **Lower horiz stiff**	Define whether horizontal stiffeners are created to the beam. If you enter one of the dimensions, stiffeners are created.	thickness = `20 mm` width = `width of the beam flange` height = `300 mm`

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

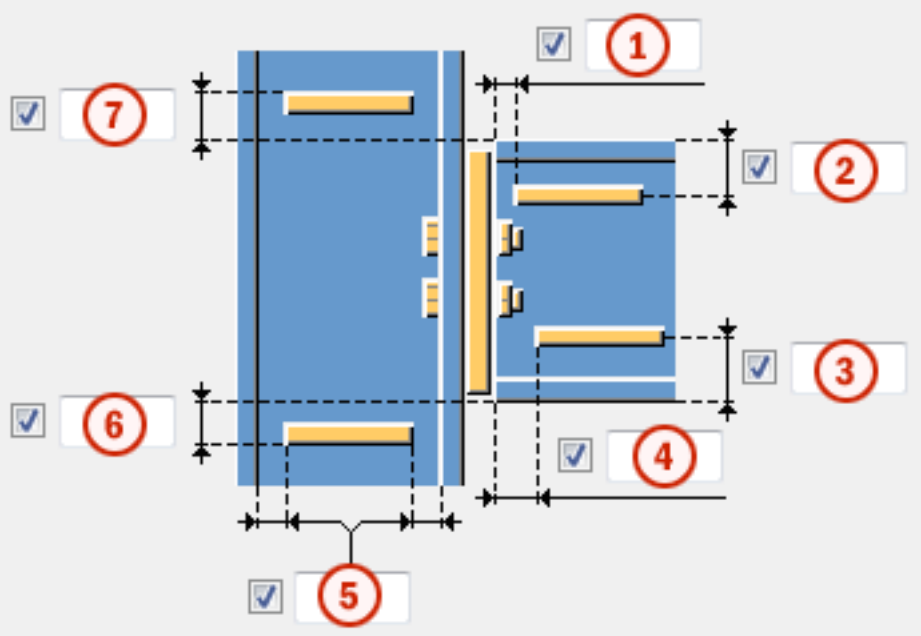
Use the **Parameters** tab to control the stiffener position and orientation.

Material of folded plate

Select the material of the bent plate. The default material is defined in the **Folded plate** box on the **Components** tab in **File menu** --> **Settings** --> **Options**.

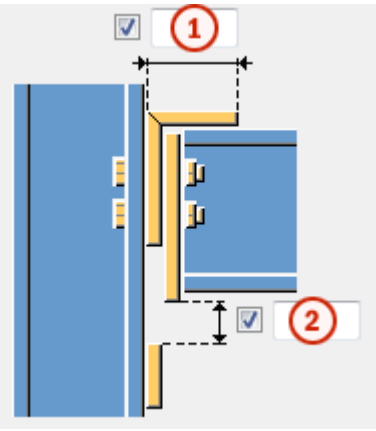
.

Stiffener positions



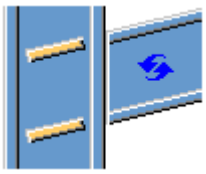
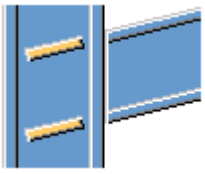
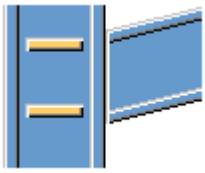
	Description	Default
1 **4**	Distance between the beam horizontal stiffener and the beam end.	
2 **3**	Distance between the beam horizontal stiffener and the beam flange.	`0.25*beam height`
5	Distance between the stiffener and the main part flange.	
6 **7**	Distance between the main part stiffener and the beam flange.	

Bent shim plate and seat length

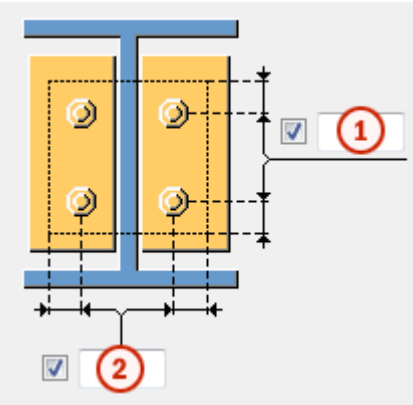


	Description	Default
1	Length of the horizontal part of the bent shim plate.	
2	Distance between the end plate and the seat. When using seat angles, change either this value or the size of the end plate to avoid collision between the secondary beam and the angle.	20 mm

Stiffener orientation

Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are parallel to the secondary part.
	Stiffeners are perpendicular to the main part.

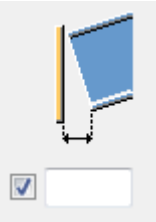
Bolt edge distances in shim plate



Define the bolt edge distances for shim plates. When these fields are empty, shim plates are of the same size as the end plate.

	Description
1	Vertical bolt edge distance in the shim plate. The size of the shim plate is defined by the bolt group and the edge distances. Vertical and horizontal edge distance define the distance symmetrically. Positive dimension increases the shim plate size.
2	Horizontal bolt edge distance in the shim plate. The size of the shim plate is defined by the bolt group and the edge distances. Vertical and horizontal edge distance define the distance symmetrically. Positive dimension increases the shim plate size.

Gap size to end plate



Define the limit value for the gap between the end plate and the secondary or main part. Use this gap when the beam is slightly curved or sloped to decide if the end angle is so small that the beam end can be straight.

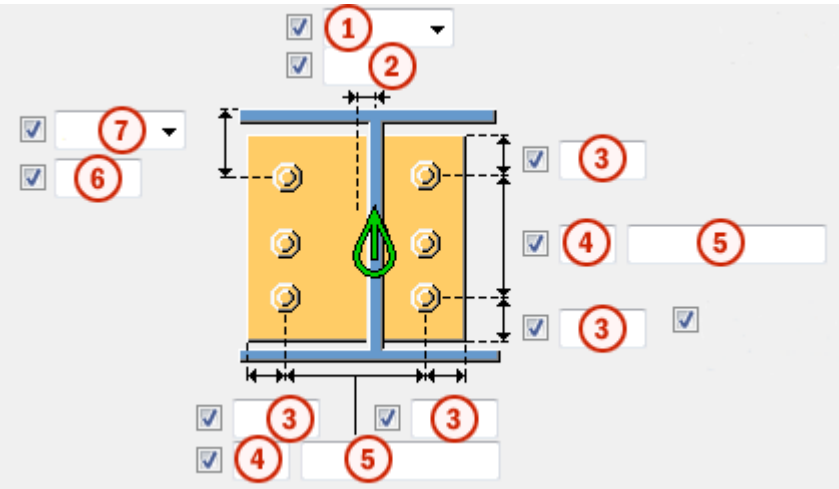
If the actual gap is smaller than this value, the beam end is left straight.

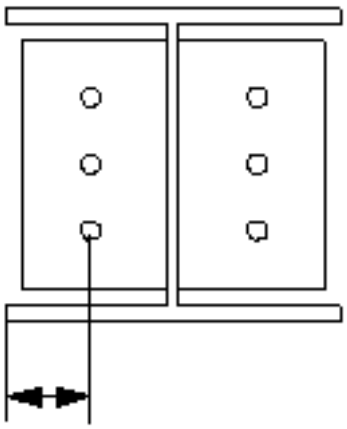
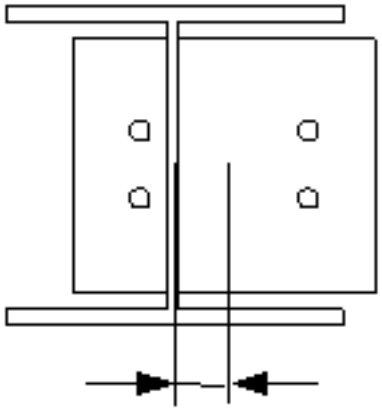
If the actual gap is larger than this value, the beam end is fitted to the end plate.

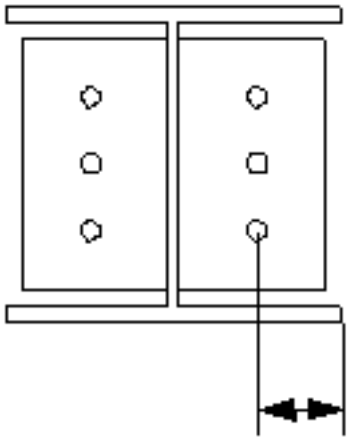
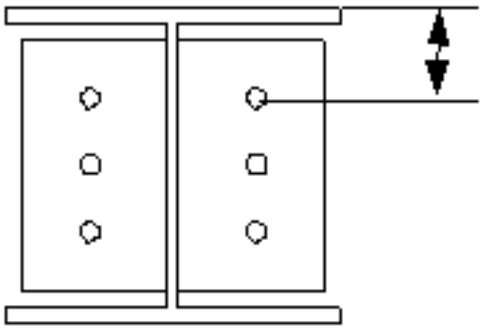
Bolts tab

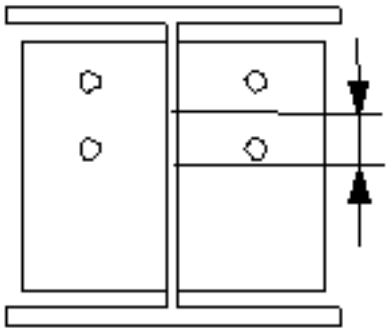
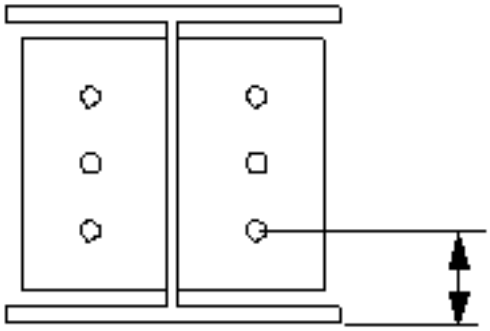
Use the **Bolts** tab to control the properties of the bolts.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt.  • **Middle**: From the center line of the secondary part to the center line of the bolts. 

	Description
	• **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Dimension for vertical bolt group position.
7	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3

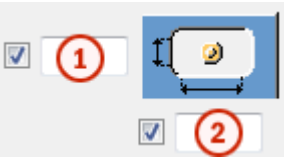
Option	Description
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



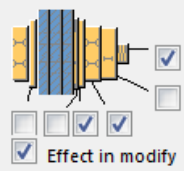
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

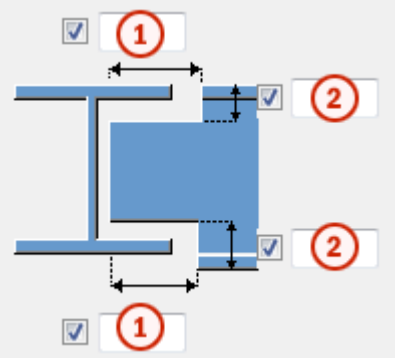
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Notch tab

Use the **Notch** tab to modify the horizontal and vertical cuts.

Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	10 mm
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest 5 mm.

General tab

Click the link below to find out more:

General tab

Design tab

Design tab

Analysis tab

Analysis tab

Welds

Click the link below to find out more:

Seating cap (37)

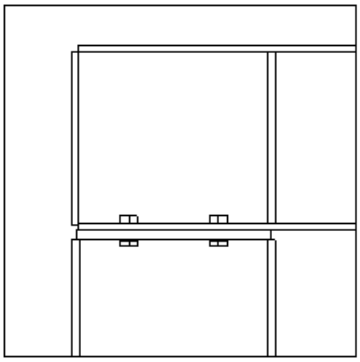
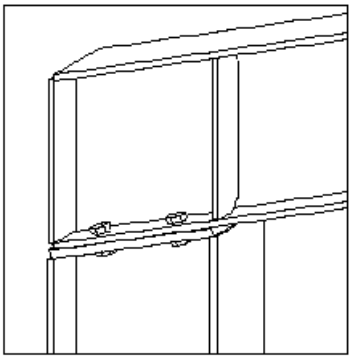
Seating cap (37) connects a beam to a column head with an end plate welded to the column head and bolted to the lower flange of the secondary beam. The

connection also creates stiffeners to the secondary part web, and optionally rectangular washer plates.

Objects created

- End plates
- Stiffeners
- Washer plates
- Bolts
- Welds

Use for

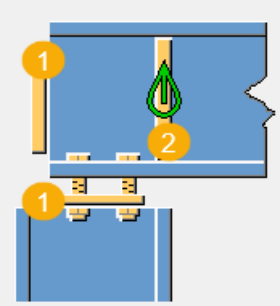
Option	Description
	Square beam connected to a column with a bolted end plate.
	Sloped beam connected to a column with a bolted end plate.

Selection order

1. Select the main part (column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

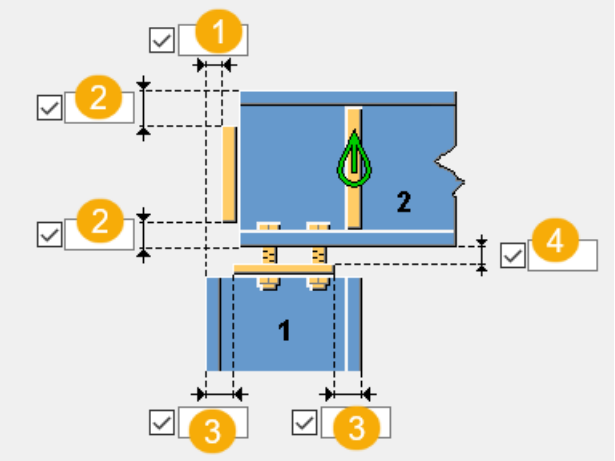


1	End plate
2	Stiffener

Picture tab

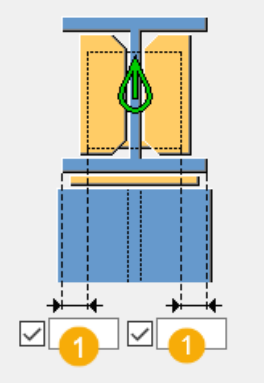
Use the **Picture** tab to define the geometry of the connection.

End plate positions



	Description	Default
1	End plate edge distance to the main part flange.	
2	End plate edge distance from the secondary part flange. You can control the column end plate size in the secondary part axis direction. Positive values move the end plate edge towards the centre.	-10 mm
3	End plate edge distance to the main part flange.	
4	Distance between the end plate and the secondary part.	

Stiffener positions



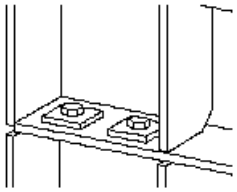
	Description	Default
1	Distance between the stiffener plate and the secondary beam flange.	0

Parts tab

Use the **Parts** tab to define the part properties.

Plates

Option	Description	Default
Prim end plate	Thickness of the main part end plate.	Half of the screw diameter The width is defined by the bolt group horizontal edge distances. The height is defined by the plate edge distances from the left and right edge of the column.
Sec stiffeners	Thickness, width and height of the stiffeners.	The default value for the stiffener thickness is 1.5*secondary beam web thickness rounded upwards to: 8, 10, 12, 16, 20, 25, 30, 35, 40, 45, and so on. If you do not define the width, stiffener width is based on the flange width. The height is equal to the distance between

Option	Description	Default
		the secondary beam flanges.
Sec end plate	Thickness of the secondary part end plate.	The size of the plate is based on the dimensions of the secondary part. You can control the dimensions on the **Picture** tab. 1.5* secondary beam web thickness rounded upwards to: 8, 10, 12, 16, 20, 25, 30, 35, 40, 45, and so on.
Washer plates	Thickness, width and height of the washer plates.  Washer plates are small rectangular plates used as washers between the bolt head and secondary beam flange.	If you do not enter a value for the thickness, the plates are not created.

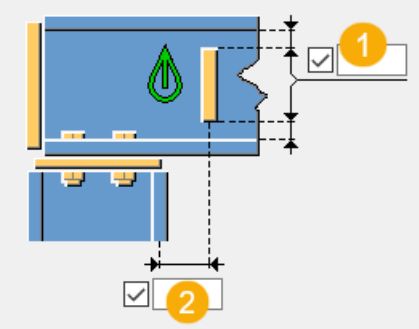
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
Name	Name that is shown in drawings and reports.	

Parameters tab

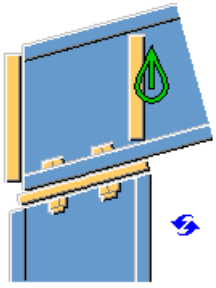
Use the **Parameters** tab to define the chamfer dimensions and stiffener position.

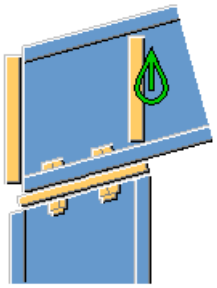
Stiffener position



	Description
1	Define the stiffener offset from the secondary part web.
2	Define the stiffener offset from the main part flange. By default, stiffeners are placed to the same plane as the column flange. Positive offset values move stiffeners to the right and negative to the left.

Stiffener angle

Option	Description
	Default Stiffeners are created parallel to the column flange. AutoDefaults can change this option.

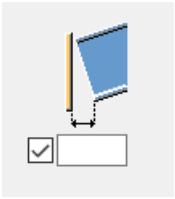
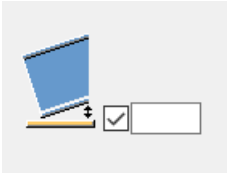
Option	Description
	Stiffeners are created parallel to the column flange.
	Stiffeners are created perpendicular to the secondary beam flange.

Gap size

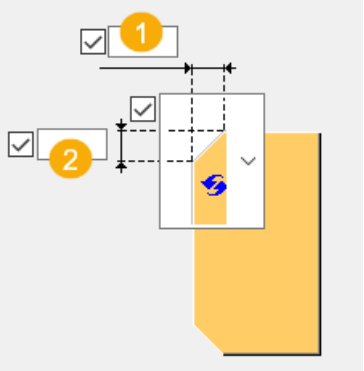
Define the limit value for the gap between the end plate and the secondary beam. Use this when the beam is slightly curved or sloped to decide if the end angle is so small that the beam end can be straight.

If the actual gap is smaller than this value, the end of the beam is left straight.

If the actual gap is larger than this value, the end of the beam is fitted to the end plate.

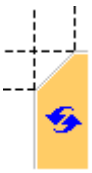
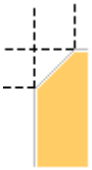
Option	Description
	Horizontal gap dimension
	Vertical gap dimension

Chamfer size



	Description
1	Horizontal chamfer dimension
2	Vertical chamfer dimension

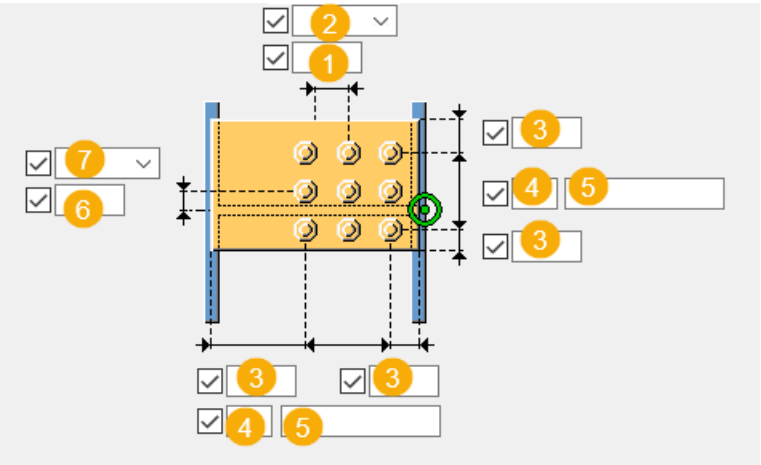
Chamfer type

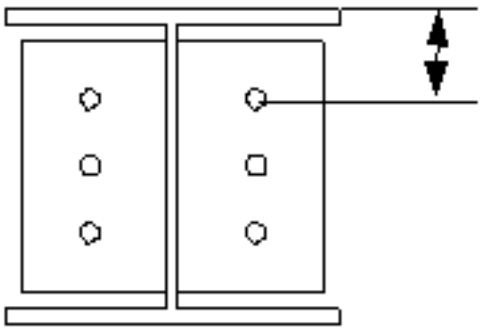
Option	Description
	Default. Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

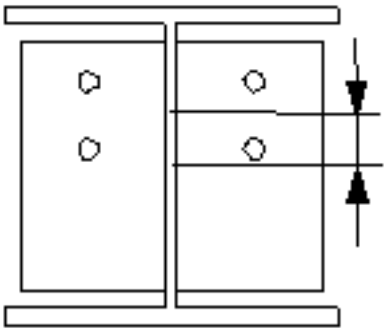
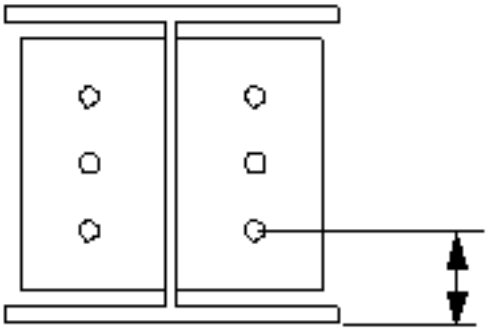
Bolts tab

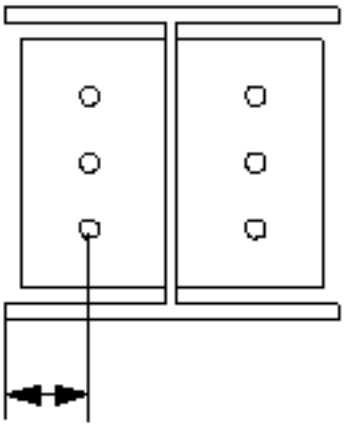
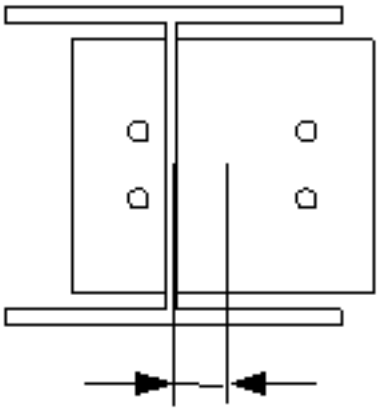
Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

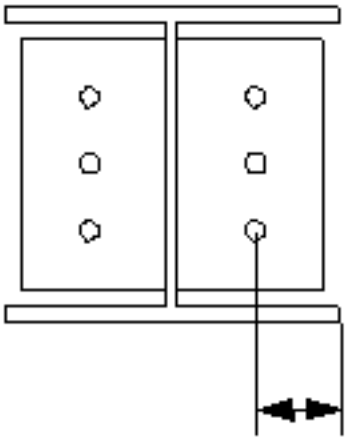
Bolt group dimensions



	Description
1	Dimension for vertical bolt group position.
2	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Dimension for horizontal bolt group position.

	Description
7	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt.  • **Middle**: From the center line of the secondary part to the center line of the bolts. 

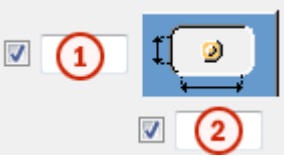
	Description
	• **Right**: From the right edge of the secondary part to the rightmost bolt. 

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

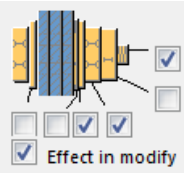


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

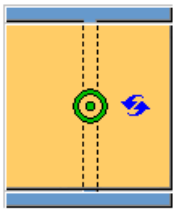
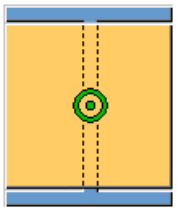
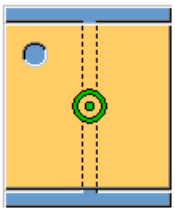
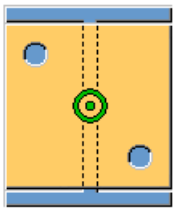
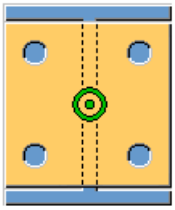
Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Holes tab

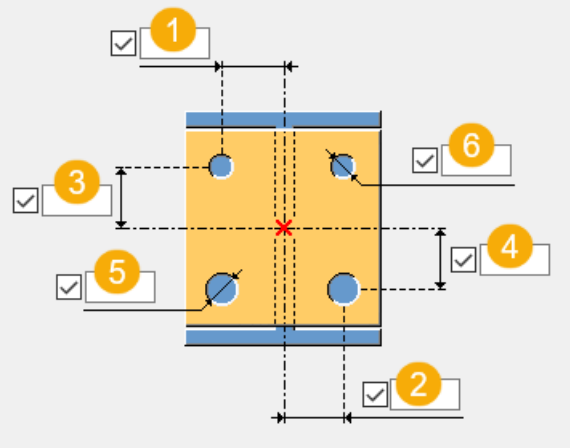
Use the **Holes** tab to define the holes created in the end plates.

Option	Description
Bolt standard	Select the bolt standard.
Bolt type	Select the bolt type to define the location where the bolts should be attached.
Read data from	You can select to use the `sinkholes.dat` definition file to specify the default values for horizontal and vertical offsets, and the diameters for upper and lower holes. The file is searched in the following order: Environment common system steel folder (`..\Environments\common\system\Steel`), model folder, `XS_FIRM`, `XS_PROJECT` and `XS_SYSTEM` folder. You can also select to define the holes in the component dialog box.

Number of holes

Option	Description
	Default No holes AutoDefaults can change this option.
	No holes
	1 hole
	2 holes
	4 holes

Hole positions



	Description
1	Horizontal distance between the end plate center and the upper hole.
2	Horizontal distance between the end plate center and the lower hole.
3	Vertical distance between the end plate center and the upper hole.
4	Vertical distance between the end plate center and the lower hole.
5	Diameter of the lower hole.
6	Diameter of the upper hole.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

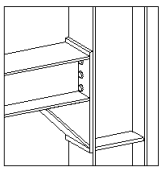
Haunch (40)

Haunch (40) connects a beam to a column using a welded haunch, or a profile haunch and a bolted end plate.

Objects created

- End plate
- Top plate
- Stiffeners
- Cap plate
- Web plate
- Haunch plates
- Compression flange plates
- Bolts
- Welds

Use for

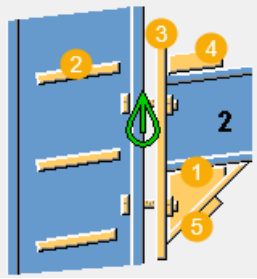
Situation	Description
	Beam connected to a column using a profile haunch and a bolted end plate.

Selection order

1. Select the main part (column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

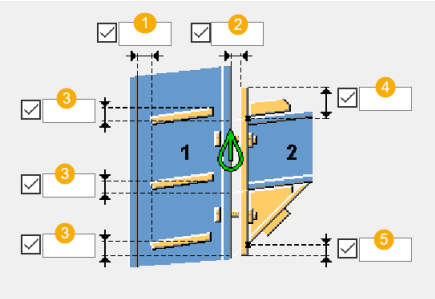


	Description
1	Haunch plate
2	Stiffener
3	End plate
4	Top plate
5	Compression flange plate

Picture tab

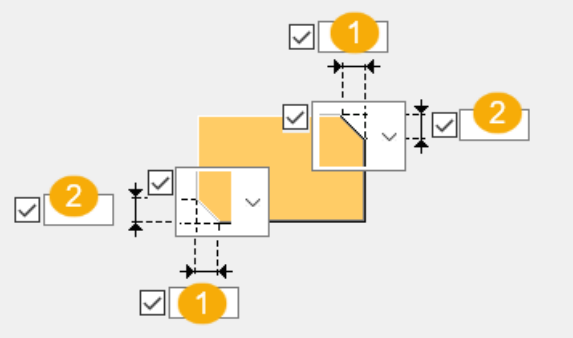
Use the **Picture** tab to define the connection dimensions, chamfers, double haunches, and rotation point.

Dimensions



	Description
1	Tolerance dimension from the stiffener edge to the column web.
2	Gap dimension between the column and the end plate.
3	Vertical stiffener dimension from the creation point.
4	End plate top position from the top of the secondary beam.
5	End plate bottom position from the haunch plate lower edge.

Top plate chamfers

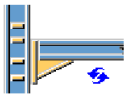
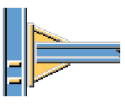
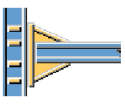


	Description
1	Horizontal chamfer dimension.
2	Vertical chamfer dimension.

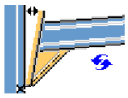
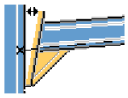
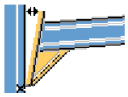
Option	Option	Description
		No chamfer
		Line chamfer
		Convex chamfer
		Concave chamfer

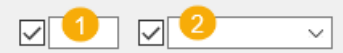
Horizontal rafter double haunch

You can select to create a double haunch if the beam is horizontal. All the plates welded to the lower haunch are mirrored on the upper haunch and the end plate is extended.

Option	Description
	Default Lower haunch is created. AutoDefaults can change this option.
	Lower haunch is created.
	Double haunch is created.
	Double haunch with column stiffeners is created.

Select rotation point

Option	Description
	Default Rotation point is in the middle of the beam lower flange. AutoDefaults can change this option.
	Rotation point is in the middle of the haunch flange.
	Rotation point is in the middle of the beam lower flange.



	Description
1	Define the tolerance value between the column and the end plate. The maximum width is at the top of the end plate. There is no tolerance at the lowest point of the end plate.
2	Select whether the tolerance is defined as a distance or as a rotation angle (in degrees).

Parameters tab

Use the **Parameters** tab to define the part properties and to control stiffener creation.

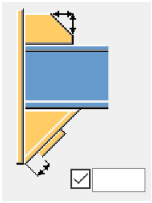
Parts

Option	Description
End plate	Thickness, width, and height of the end plate.
Add. stiffener	Thickness, width, and height of the additional stiffener.
Rafter bottom stiffener	Thickness, width, and height of the beam bottom stiffener.
Compr. stiffener	Thickness, width, and height of the compression stiffener.
Top plate	Thickness, width, and height of the top plate.
Cap plate	Thickness, width, and height of the cap plate.

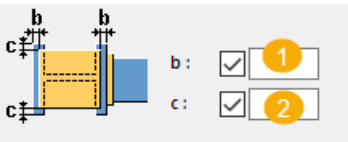
Option	Description
Diagonal stiffener	Thickness and width of the diagonal stiffener.
Beam stiffener	Thickness, width, and height of the beam stiffener.
Morris stiffener	Thickness and width of the Morris stiffener.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Compression flange plate position

Option	Description
	Compression flange plate edge distance to the end plate bottom edge. If the haunch is not created, the plate is positioned on the beam flange.

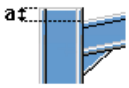
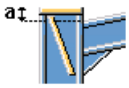
Cap plate position



	Description	Default
1	Cap plate edge distance from the column flange.	5 mm The cap plate is by default 5 mm smaller than the profile.
2	Cap plate edge distance from the column flange. Define this dimension when the cap plate width has not been defined.	5 mm

Option	Description
	Default Cap plate is created parallel with the beam. AutoDefaults can change this option.
	Cap plate is created parallel with the beam.
	Cap plate is horizontal.

Additional stiffener

Option	Description
	Default Cap plate or stiffener is not created. AutoDefaults can change this option.
	Cap plate or stiffener is not created.
	Cap plate or stiffener is not created. Define the vertical dimension for moving the column end fitting plane.
	Cap plate and stiffener are created. Define the vertical dimension for moving the cap plate or the column end fitting plane.
	Horizontal stiffener is created.

Option	Description
	Stiffener is created parallel with the skewed beam.

Bottom flange stiffener

Option	Description
	Default Stiffener is not created. AutoDefaults can change this option.
	Horizontal stiffener is created.
	Stiffener is not created.
	Skewed stiffener is created.

Morris stiffener

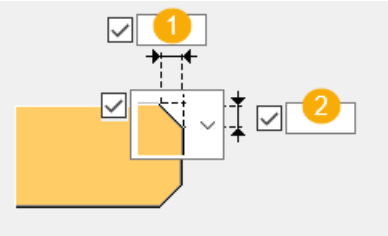
If a haunch is not created, the Morris stiffener is not created either.

Option	Description
	Default Horizontal stiffener is created. Define the upper stiffener dimension from the column web. AutoDefaults can change this option.
	Horizontal stiffener is created. Define the upper stiffener dimension from the column web.
	Stiffener is not created.
	Skewed stiffener is created. Define the upper stiffener dimension from the column web.

Stiffener chamfer shape and dimensions

Option	Description
	No chamfer

Option	Description
	Line chamfer
	Convex chamfer
	Concave chamfer



	Description
1	Horizontal chamfer dimension.
2	Vertical chamfer dimension.

Beam stiffener

Option	Description
	Default Vertical stiffener in the direction of the column. AutoDefaults can change this option.
	Vertical stiffener in the direction of the column.
	Stiffener is created perpendicular to the beam axis.

Beam stiffener offset

Option	Description
	Beam stiffener offset from the default position where the beam stiffener is created. The default position is in the place where the haunch profile flange touches the secondary beam.

Extra length for haunch flange plate

Define the distance that extends the flange plate of a welded haunch.

Parameter to avoid solid error

Define the distance value to avoid solid errors. If a haunch profile is selected, and a chamfer is created in the profile, the haunch profile may disappear. You can solve this issue by increasing this value. The default value is `0.5 mm`.

Haunch tab

Use the **Haunch** tab to define the haunch plate properties, type, and geometry.

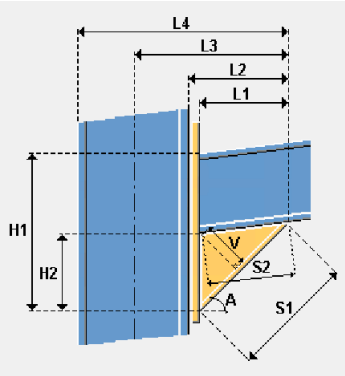
Parts

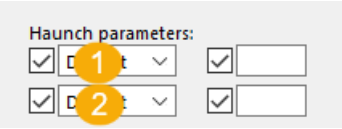
Option	Description
Haunch prof	In **Haunch type**, select **Profile** or **Default**. Select the haunch profile from the profile catalog.
Vert haunch pl	In **Haunch type**, select **Welded plates**. Thickness of the haunch plate.
Horiz haunch pl	In **Haunch type**, select **Welded plates**. Thickness and width of the haunch plate.
Closure plate	Thickness and height of the closure plate.
Comp. flange pl	Thickness, width, and height of the flange plate.
Haunch prolongation	If the connection is created with an open beam, the beam bottom flange can be extended to reach the end plate. Select to extend the beam bottom flange on the **Open beam** tab.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Haunch parameters





	Description	Default
1	Select the first dimension for the haunch geometry. If the selected dimension cannot be used for the geometry, a red component symbol is displayed. For profile hauches: If only one dimension is selected (first or second), a symmetrical haunch is created using the symmetrical cut for profile haunch options.	**A**, **L1** dimension where L1 = 1/5 from the beam length. Profile haunch: a symmetrical haunch using the symmetrical cut for profile haunch options.
2	Select the second dimension for the haunch geometry. If you select two dimensions that cannot be used for the geometry, a red component symbol is displayed. For profile haunches: If only one dimension is selected (first or second), a	**A**, **L1** dimension where L1 = 1/5 from the beam length. Profile haunch: a symmetrical haunch using the symmetrical

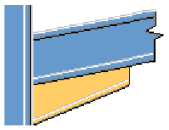
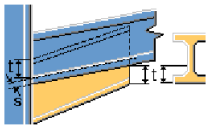
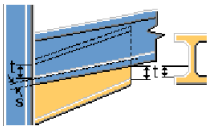
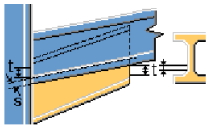
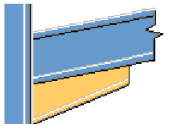
	Description	Default
	symmetrical haunch is created using the symmetrical cut for profile haunch options.	cut for profile haunch options.

Symmetrical cut for profile haunch

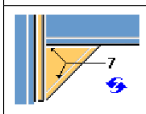
Select the calculation method, and define the tolerance value (**t**) and saw plate thickness (**s**) to create a symmetrical profile haunch. This setting is not used for welded plates.

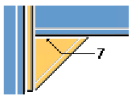
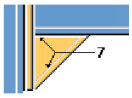
When measured, the tolerance distance might have an error between 0 mm – 2 mm, resulting from the approximations done during calculation. Check that the result is as intended. Note that if you define the vertical cut distance at the upper end of the hauch, the tolerance value is not used.

With saw plate thickness, you can control how much of the material will be wasted because of the sawing.

Option	Description
	Default Inner haunch flange to outer beam flange calculation. AutoDefaults can change this option.
	Define the tolerance value from the outer haunch plate flange, and the saw plate thickness.
	Define the tolerance value from the inner haunch plate flange, and the saw plate thickness.
	Define the tolerance value from the haunch plate web, and the saw plate thickness.
	Inner haunch flange to outer beam flange calculation.

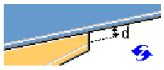
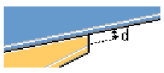
Welds in the vertical haunch plate

Option	Description
	Default Both welds are created. AutoDefaults can change this option.

Option	Description
	Weld is created on the secondary beam.
	Both welds are created.

Cut at the upper end of the haunch

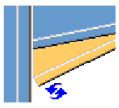
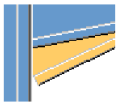
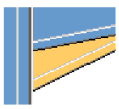
Using this option you can select if the haunch is created as fitted to the secondary beam, or if there should be some material left, as specified with the **d** option.

Option	Description
	Default Vertical cut at the upper end of the haunch. Define the cut distance. AutoDefaults can change this option.
	The cut is perpendicular to the beam flange.
	Vertical cut at the upper end of the haunch. Define the cut distance.
	Haunch is not cut.

Haunch end cuts parallel (only for profiles)

Select **Yes** to create the haunch cut at the upper end and the haunch cut at the lower end parallel to each other.

Cut at the lower end of the haunch (only for profile haunch)

Option	Description
	Default Vertical cut at the lower end of the haunch. AutoDefaults can change this option.
	Vertical cut at the lower end of the haunch.
	Haunch is not cut.

Extra plates

Use the **Extra plates** tab to define the web plate and bolt plate properties.

Parts

Option	Descripton
Web plate	Thickness, width, and height of the web plate.
Upper bolt plate	Thickness and width of the upper bolt plate.
Lower bolt plate	Thickness and width of the lower bolt plate.
Plate on column	Thickness and width of the plate on the column.

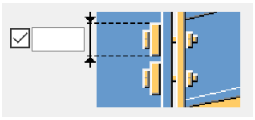
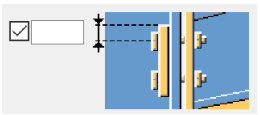
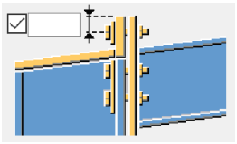
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Bolt plate

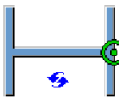
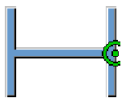
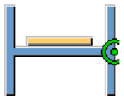
Option	Description
	Default One bolt plate for each bolt in the group. AutoDefaults can change this option.
	One bolt plate for each bolt in the group.

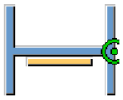
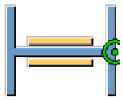
Option	Description
	One bolt plate for each column of bolts in the group.
	One bolt plate for all the bolts in the group.
	Bolt plate is not created.

Bolt plate dimensions

Option	Description
	Define the bolt plate height when creating one bolt plate for each bolt in the group.
	Define the vertical bolt plate edge distance for the first and last bolt in the group when creating one bolt plate for each column of bolts, or one bolt plate for all the bolts in the group.
	Define the thickness of the horizontal bolt plate that continues the column flange. If the first bolt in the beam bolt group is positioned above the beam – column collision area, a new bolt plate is created for the first bolt row in the group.

Create web plate

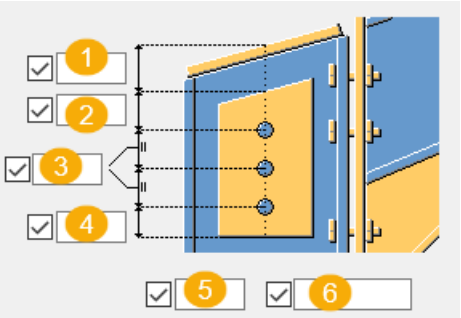
Option	Description
	Default Web plate is not created. AutoDefaults can change this option.
	Web plate is not created.
	One web plate is created on the right side of the column web.

Option	Description
	One web plate is created on the left side of the column web.
	Two web plates are created.

Web plate settings

Option	Description	Default
Web plate edge type	Select how the web plate is cut at the higher end. **Bevel** cuts the web plate at the same angle the column cap plate is positioned. If the column cap plate is horizontal or does not exist, the web plate is cut **Square**.	**Square**
Number of holes	Define the number of holes in the web plate.	2
Hole diameter	Define the diameter of the holes in the web plate.	20 mm
Horizontal, **Vertical**	Define the web plate welding size: • Define the left and right weld between the column and the web plate. • Define the lower vertical edge for the web plate.	5 mm

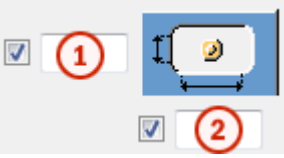
Web plate dimensions



Option	Description	Default
1	Distance from the edge of the extra plate to the top of the cap plate.	`5 mm`
2	Hole edge distance of the first hole in the web plate.	`30 mm + hole_diameter/2`
3	Vertical spacing between the holes.	`160 mm`
4	Vertical hole edge distance to the lower edge of the web plate.	`30 mm + hole_diameter/2`
5	Number of holes in the columns.	`1`
6	Distance between the holes in the columns.	`hole_diameter x 2`

Slotted holes

You can define slotted or oversized holes.



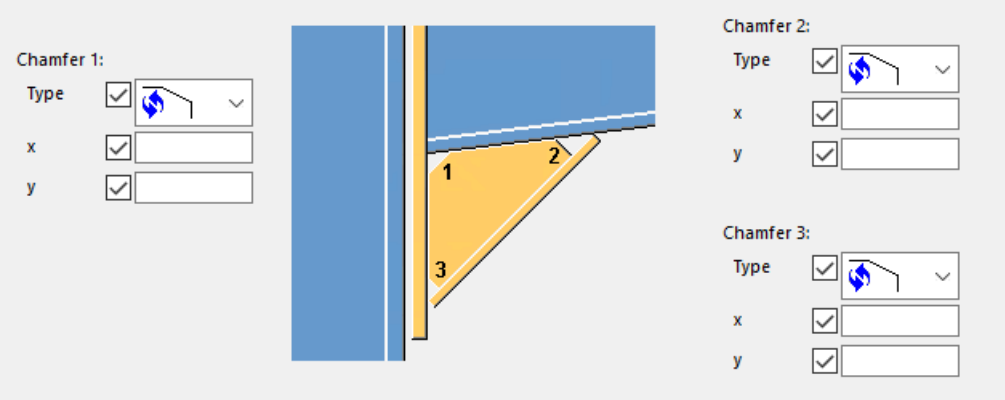
	Description
1	Vertical dimension of the slotted hole.
2	Horizontal dimension of the slotted hole, or allowance for oversized holes.

Chamfers tab

Use the **Chamfers** tab to define the chamfer type and dimensions.

Chamfers

Define the chamfer type and dimensions separately for each corner.



Chamfer	Description	Default
Chamfer 1	Select the chamfer type and define the dimensions. This chamfer is available for both profile and welded plate haunches.	No chamfer
Chamfer 2	Select the chamfer type and define the dimensions. This chamfer is available for welded plate haunches only.	No chamfer
Chamfer 3	Select the chamfer type and define the dimensions. This chamfer is available for welded plate haunches only.	No chamfer

Holes tab

Use the **Holes** tab to control the galvanizing holes in the end plate.

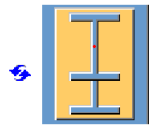
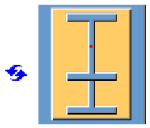
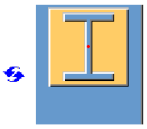
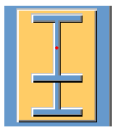
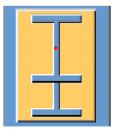
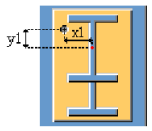
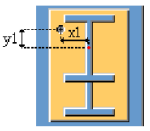
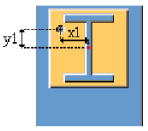
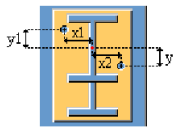
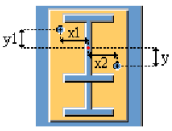
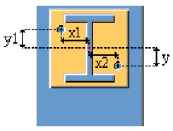
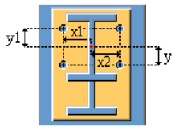
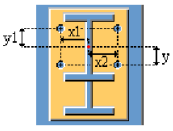
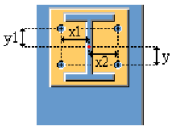
Option	Description
Bolt standard	Select the bolt standard.
Bolt type	Select the bolt type to define the location where the bolts should be attached.
Read data from	You can select to use the `sinkholes.dat` definition file to specify the default values for horizontal and vertical offsets, and

Option	Description
	the diameters for upper and lower holes. The file is searched in the following order: Environment common system steel folder (`..\Environments\common\system\Steel`), model folder, `XS_FIRM`, `XS_PROJECT` and `XS_SYSTEM` folder. You can also select to define the holes in the component dialog box.

Hole positions and number of holes

Define the hole positions and the number of holes in the end plate and the cap plate.

The hole group center in the end plate is the middle point of the rafter and the middle point of the haunch, if the haunch exists. The hole group center in the cap plate is the middle point of the column. The hole groups are composed of 0, 1, 2, or 4 holes.

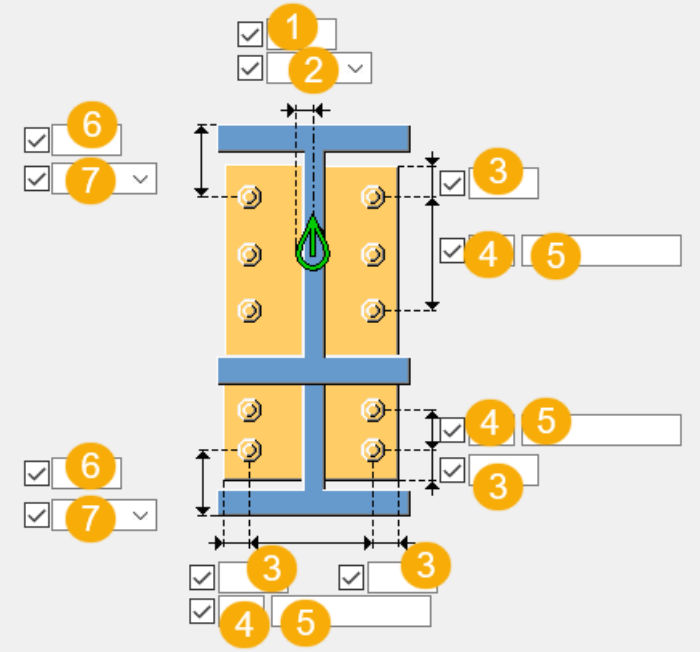
Holes group in rafter	Holes group in haunch	Holes group in cap plate
		
		
		
		
		

	Description	Default
D1	Define the hole diameter for the holes positioned by the x1 and y1 dimensions.	20 mm
D2	Define the hole diameter for the holes positioned by the x2 and y2 dimensions.	20 mm
x1	Define the x1 position of the galvanization holes.	0
y1	Define the y1 position of the galvanization holes.	0
x2	Define the x2 position of the galvanization holes.	0
y2	Define the y2 position of the galvanization holes.	0

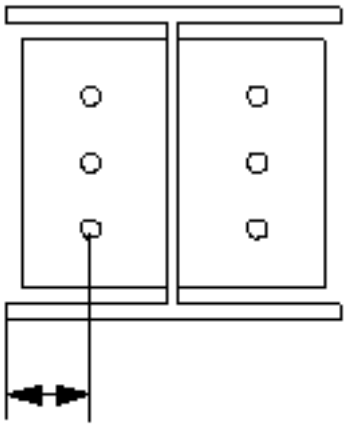
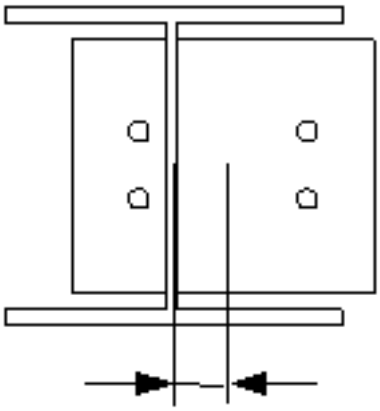
Bolts tab

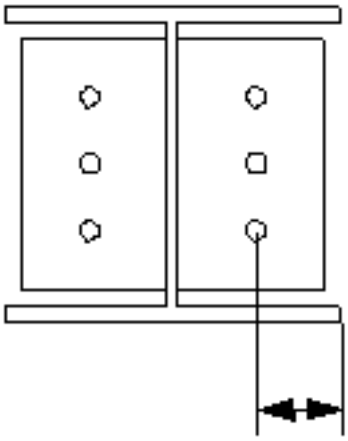
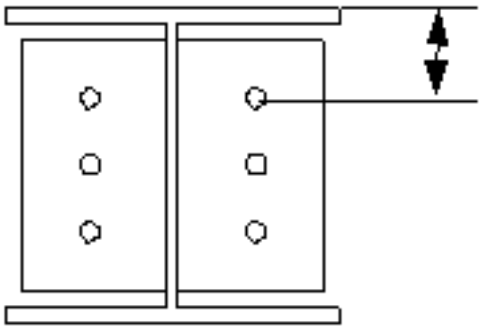
Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

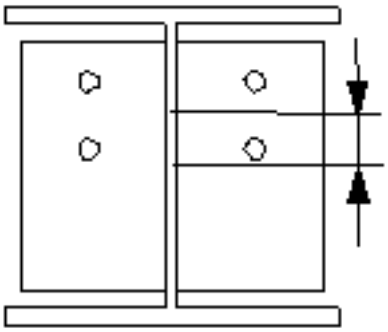
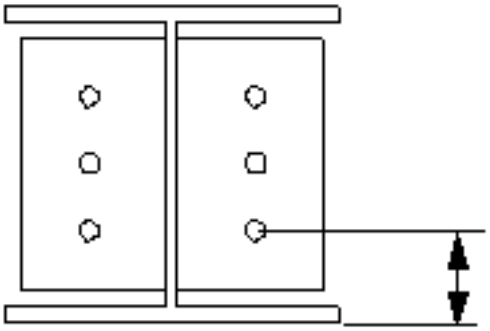
Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.

	Description
2	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt.  • **Middle**: From the center line of the secondary part to the center line of the bolts. 

	Description
	• **Right**: From the right edge of the secondary part to the rightmost bolt. 
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Dimension for vertical bolt group position.
7	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

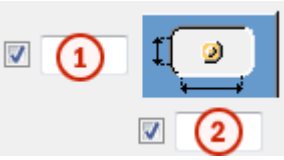
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

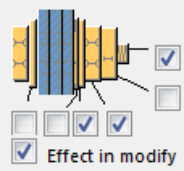


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

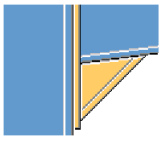
Bolting direction

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1
	Bolting direction 2

Open beam tab

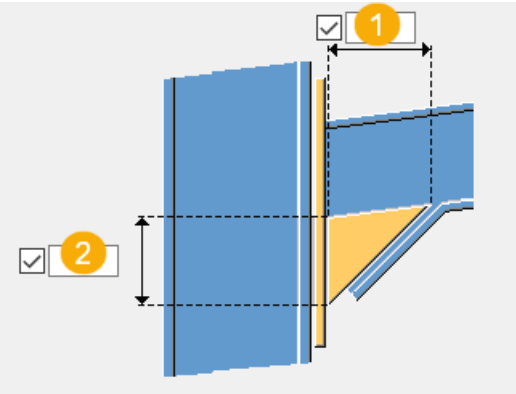
Use the **Open beam** tab to create an open beam haunch. To create the haunch, the beam web is cut and the profile is opened at a desired angle. A welded plate is placed as the haunch web.

Create open beam

Option	Description
	Open beam is not created. This is the default.

Option	Description
	Open beam is created. If you create an open beam, the options on the **Haunch** tab are ignored.

Haunch dimensions



	Description	Default
1	Horizontal haunch dimension	300 mm
2	Vertical haunch dimension	300 mm

Beam extension

Option	Description
	Beam bottom flange is not extended. This is the default.
	Beam bottom flange is extended to reach the end plate.

Haunch part to the beam

Option	Description
	The haunch is added to the beam.

Option	Description
	The haunch is an independent part that is welded to the beam. This is the default.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

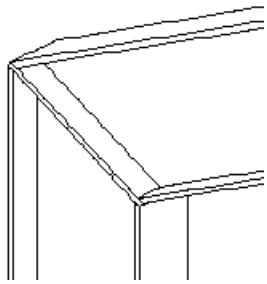
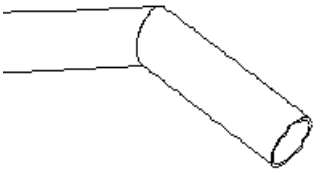
Cranked beam (41)

Cranked beam (41) connects two beam ends. The beam ends are fitted to an angle which is an average of the beam end angles. A plate can be created between the connected parts.

Objects created

- End plates
- Haunch plates
- Stiffeners
- Bolts
- Welds

Use for

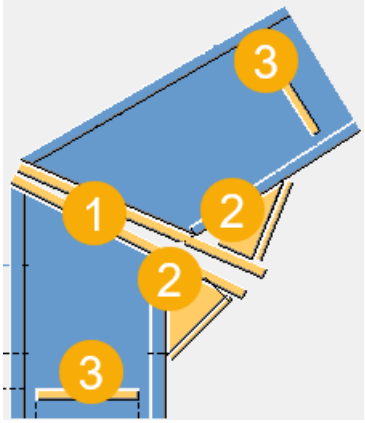
Situation	Description
	Beam ends are fitted.
	Beam ends are fitted. Tubular profiles are used.

Selection order

1. Select the main part (beam).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key



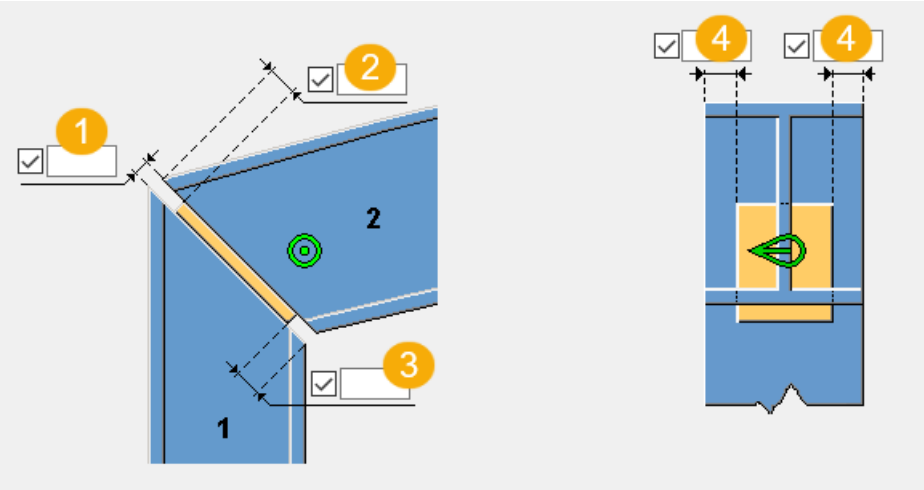
	Description
1	End plate
2	Haunch plate

	Description
3	Stiffener

Picture tab

Use the **Picture** tab to define the geometry of the connection.

Dimensions



	Description
1	Define the gap between the parts. The gap is created on both sides of the end plate.
2	Define the end plate edge distance from the top of the secondary part.
3	Define the end plate edge distance from the bottom of the main part.
4	Define the end plate edge distance to the left and right edge of the secondary part.

Parts tab

Use the **Parts** tab to define the properties of the end plates, haunch plates, and stiffeners.

Parts

Part	Description
Plate	Thickness, width and height of the end plate.
Sec plate	Thickness, width and height of the secondary end plate.

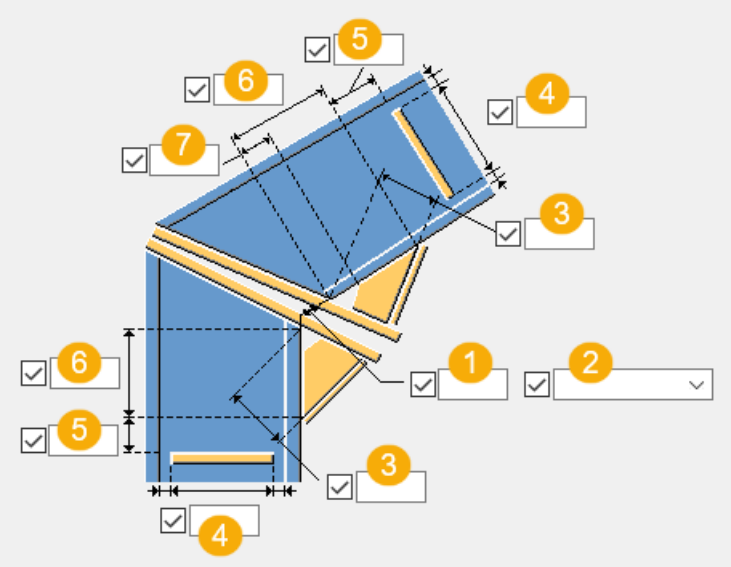
Part	Description
Haunch prof	Select the haunch profile from the profile catalog.
Vert haunch pl	Thickness of the vertical haunch plate.
Horiz haunch pl	Thickness and width of the horizontal haunch plate.
Haunch prof 2	Select the second haunch plate profile from the profile catalog.
Vert haunch pl 2	Thickness of the second vertical haunch plate.
Horiz haunch pl 2	Thickness and width of the second horizontal haunch plate.
Column stiffener	Thickness, width and height of the column stiffener.
Beam stiffener	Thickness, width and height of the column stiffener.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Parameters tab

Use the **Parameters** tab to define the plate positions and dimensions.

Plate positions and dimensions

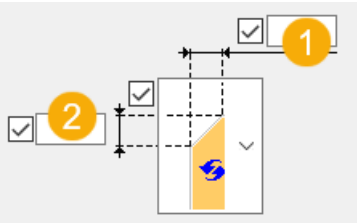


	Description
1	Gap between the end plates. You can define the gap either as a distance or as an angle.
2	Select whether the gap is defined as a distance or as an angle (degrees). By default, the gap is created as a distance.
3	Haunch plate height
4	Gap between the stiffeners and beam flanges
5	Stiffener edge distance to the haunch edge
6	Haunch plate length
7	Haunch plate chamfer size

Parallel haunch end cuts

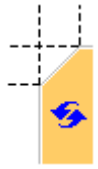
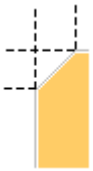
If you have selected the haunch profile from the profile catalog, you can select that the haunch ends are cut parallel.

Chamfer dimensions



1	Horizontal chamfer dimension
2	Vertical chamfer dimension

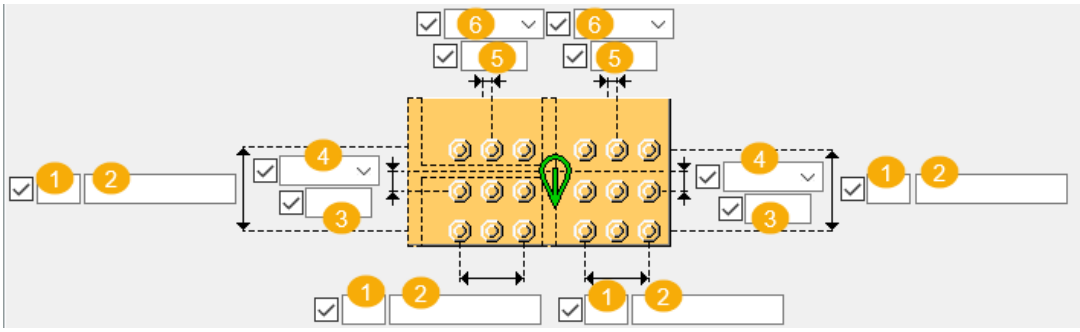
Chamfer type

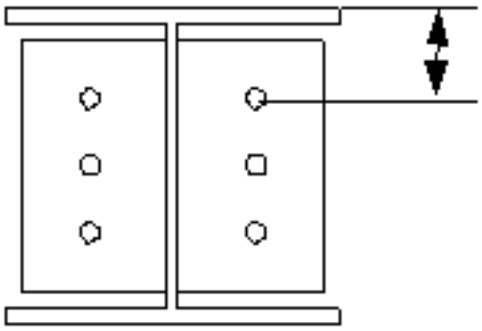
Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

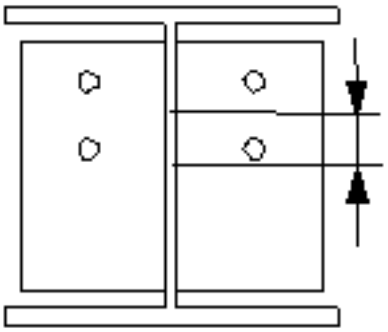
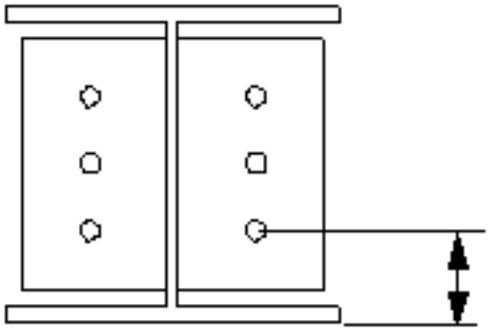
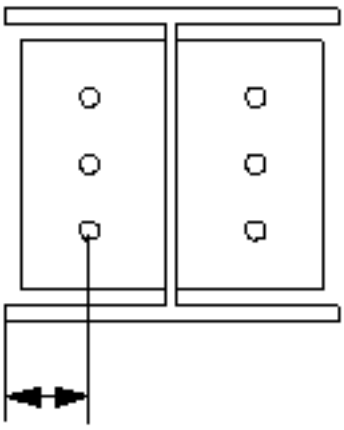
Bolts tab

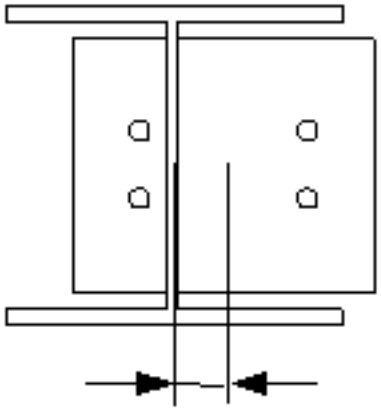
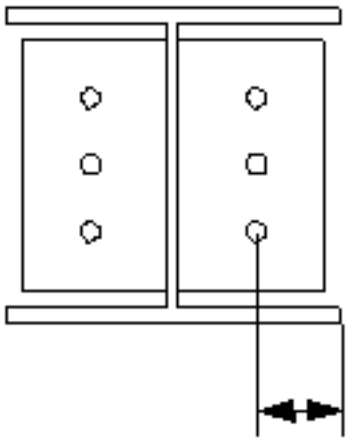
Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions



	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Dimension for vertical bolt group position.
4	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
5	Dimension for horizontal bolt group position.
6	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt. 

	Description
	• **Middle**: From the center line of the secondary part to the center line of the bolts.  • **Right**: From the right edge of the secondary part to the rightmost bolt. 

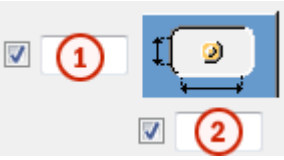
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

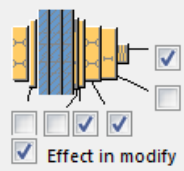


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

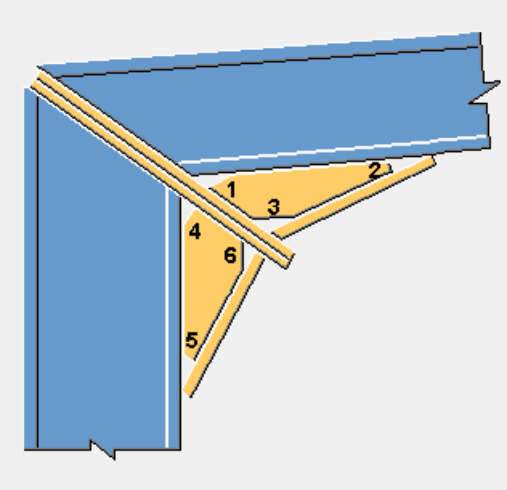
Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Chamfers tab

Use the **Chamfers** tab to define the chamfer types and dimensions for the haunch plate chamfers.

Chamfers

You can define each chamfer separately.

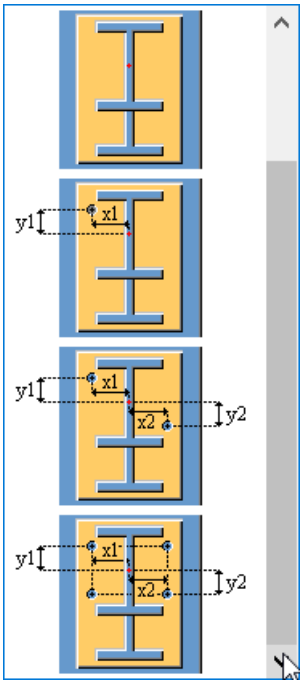


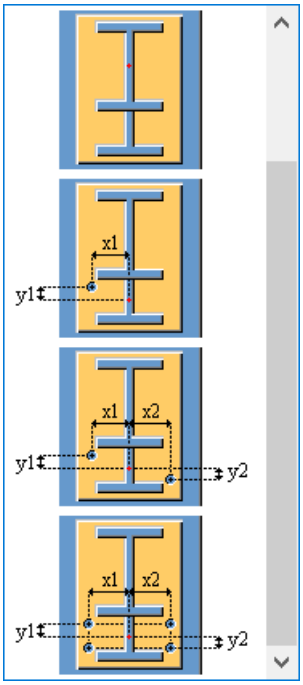
Holes tab

Use the **Holes** tab to define the holes created in the end plates.

Hole dimensions

You can define the holes separately for the main part end plate and the secondary part end plate. The main part end plate values are used as default values for the secondary part end plate.

Option	Description
	Define the hole group dimensions in the rafter.

Option	Description
	Define the hole group dimensions in the haunch.

Option	Description
Bolt standard	Select the bolt standard.
Bolt type	Select the bolt type to define the location where the bolts should be attached.
Read data from	You can select to use the `sinkholes.dat` definition file to specify the default values for horizontal and vertical offsets, and the diameters for upper and lower holes. The file is searched in the following order: Environment common system steel folder (`..\Environments\common\system\Steel`), model folder, `XS_FIRM`, `XS_PROJECT` and `XS_SYSTEM` folder. You can also select to define the holes in the component dialog box.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Create welds

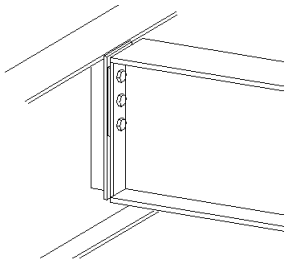
Partial stiff end plate (65)

Partial stiff end plate (65) creates a beam-to-beam end plate connection. The connection creates an end plate on the secondary beam, a stiffener plate and a connection front plate at the main beam. In addition, stiffener plates can be added to both sides, a bottom plate can be added to the bottom of the main stiffener plate, and shim plates can be created.

Objects created

- End plate
- Shear tab
- Stiffeners
- Shim plates
- Bolts
- Welds

Use for

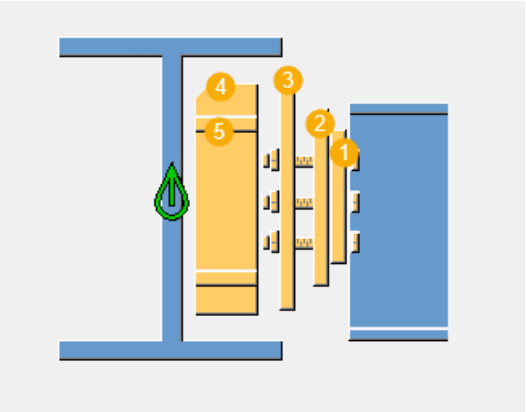
Situation	Description
	Two beams connected with bolted end plates.

Selection order

1. Select the main part (beam).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

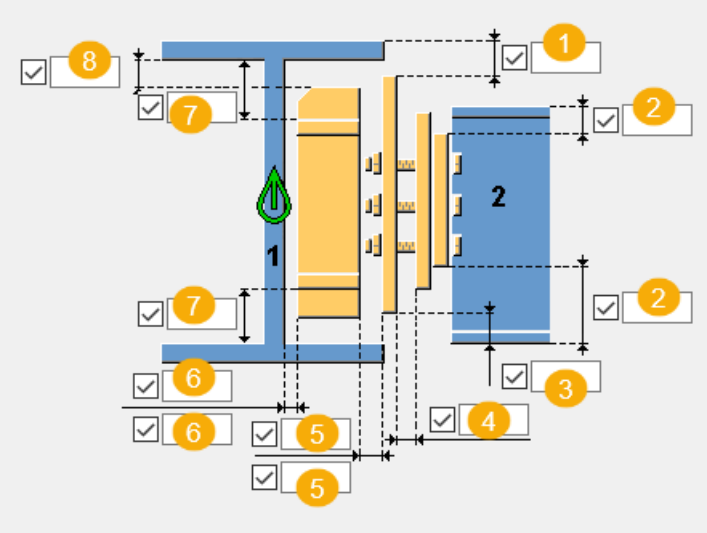


	Description
1	End plate
2	Shim plate
3	Front plate
4	Shear tab
5	Stiffener

Picture tab

Use the **Picture** tab to define the part positions.

Dimensions



	Description
1	Front plate edge distance from the main part flange.
2	End plate edge distance from the secondary part flange.
3	Front plate edge distance from the secondary part flange.
4	Gap between the front plate and the shim plate.
5	Gap between the shear tab and the front plate. The lower box is for controlling the shear tab on the other side of the main part.
6	Gap between the shear tab and the main part web. The lower box is for controlling the shear tab on the other side of the main part.
7	Stiffener edge distance from the main part web.
8	Shear tab edge distance from the main part web.

Gap size

Define the limit value for the gap between the end plate and the secondary beam. Use this when the beam is slightly curved or sloped to decide if the end angle is so small that the beam end can be straight.

If the actual gap is smaller than this value, the end of the beam is left straight.

If the actual gap is larger than this value, the end of the beam is fitted to the end plate.

Parts tab

Use the **Parts** tab to define the part properties.

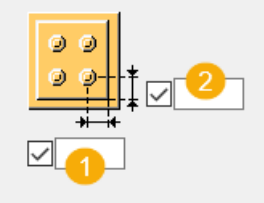
Parts

Option	Description
End plate	Thickness of the end plate.
Front plate	Thickness and width of the front plate.
Shear plate	Thickness, width and height of the end plate.
Fill shear plate width	Select whether to use the `XS_STANDARD_STIFFENER_WIDTH_TOLERANCE` advanced option for the width.
Stiffener	Thickness and width of the stiffener.
Bottom stiffener as one plate	Select whether the bottom stiffener is created as one plate.
Fitting plate 1, **Fitting plate 2**, **Fitting plate 3**	Thickness of the shim plate.
Bottom plate	Select the bottom plate profile from the profile catalog.
Profile	Select a profile. When you select a profile, this profile is created instead of the front plate.
Profile rotation	Select whether the profile is rotated horizontally or vertically.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

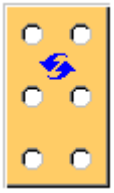
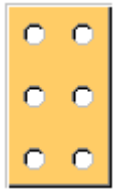
Option	Description	Default
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

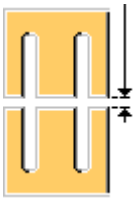
Shim plate bolt edge distance



	Description
1	Horizontal bolt edge distance in the shim plate.
2	Vertical bolt edge distance in the shim plate.

Shim plate shape

Option	Description
	Default Holes are based on the bolt group of the connection. AutoDefaults can change this option.
	Holes are based on the bolt group of the connection.
	Finger shim plate with horizontal slots. The plate can be installed from the right or the left side of the connection.
	Finger shim plate with vertical slots. The plate can be installed from the top of the connection.

Option	Description
	Two separate finger shim plates with horizontal slots.
	Two separate finger shim plates with vertical slots.

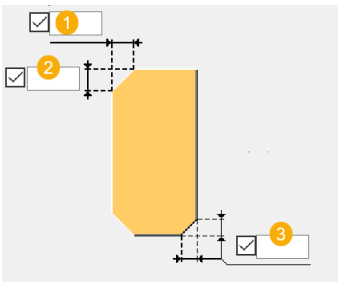
Tolerance

Define the tolerance for the slots in finger shim plates. The width of the slot is the bolt diameter + the tolerance. For two separate shim plates, also define the tolerance between the plates.

Parameters tab

Use the **Parameters** tab to define the chamfers of the shear tab and stiffeners.

Shear tab chamfer dimensions

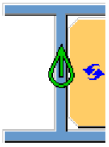
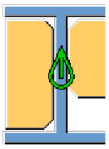


	Description	Default
1	Horizontal chamfer dimension.	30 mm
2	Vertical chamfer dimension.	30 mm
3	Bottom chamfer dimension.	30 mm

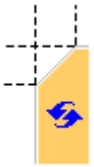
Option	Description
	Default Line chamfer AutoDefaults can change this option.

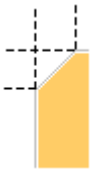
Option	Description
	No chamfer
	Line chamfer
	Convex chamfer
	Concave chamfer

Shear tab shape

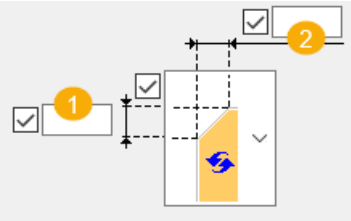
Option	Description
	Default Full Creates a full shear tab of the same height as the web of the main part. AutoDefaults can change this option.
	Full
	Shear tabs are created on both sides of the main part web.
	Partial shear tab is created on the other side of the main part web.

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.

Option	Description
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Chamfer dimensions

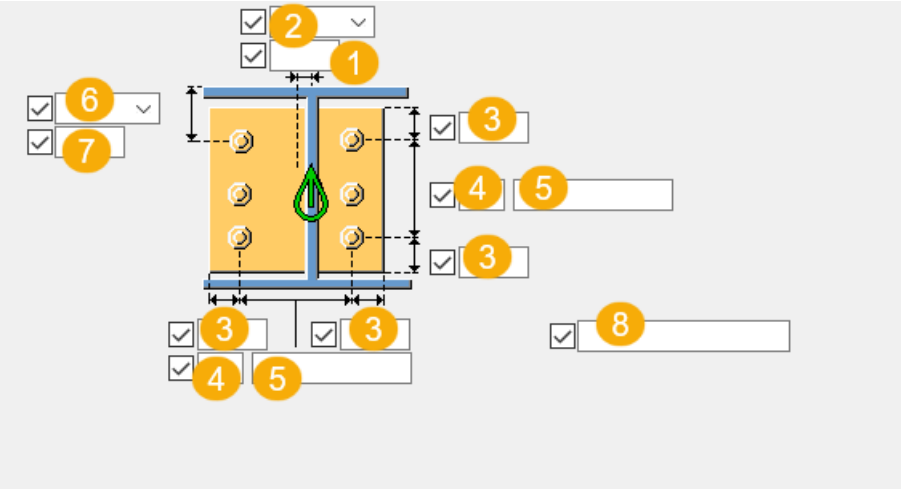


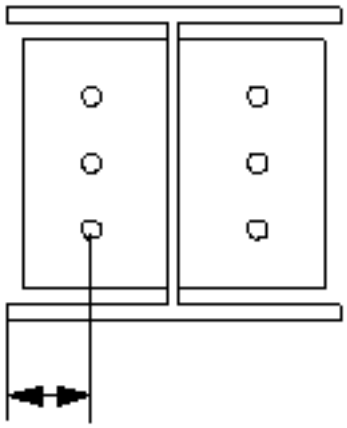
	Description	Default
1	Vertical dimension of the chamfer.	`10 mm`
2	Horizontal dimension of the chamfer.	`10 mm`

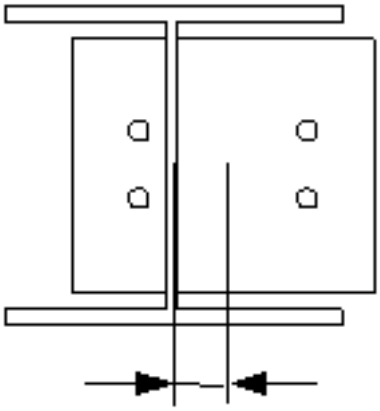
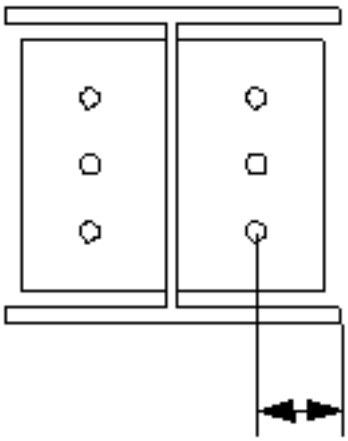
Bolts tab

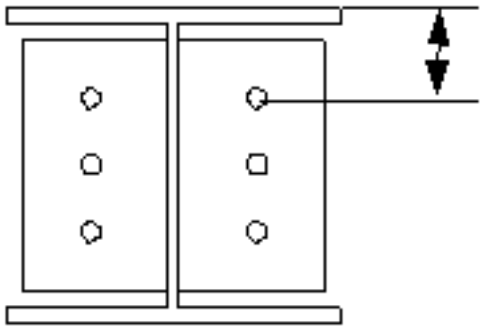
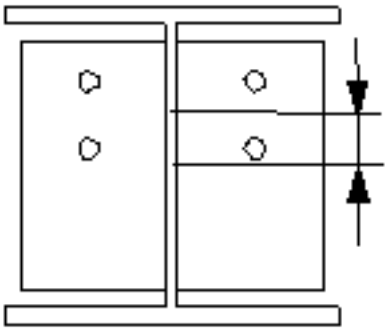
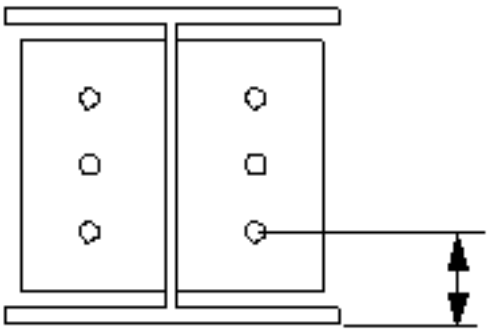
Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.
2	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt. 

	Description
	• **Middle**: From the center line of the secondary part to the center line of the bolts.  • **Right**: From the right edge of the secondary part to the rightmost bolt. 
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

	Description
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Dimension for vertical bolt group position.

	Description
8	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Bolt type

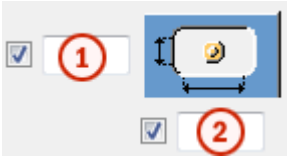
Select the bolt type to define the location where the bolts should be attached.

Bolt comment

You can define a bolt comment.

Slotted holes

You can define slotted, oversized, or tapped holes.



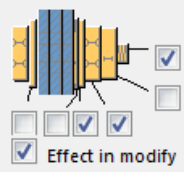
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.

Option	Description	Default
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Bolting direction

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.

Option	Description
	Bolting direction 1
	Bolting direction 2

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Holes - end plate tab

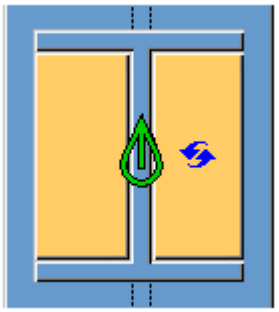
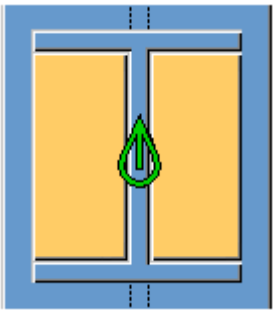
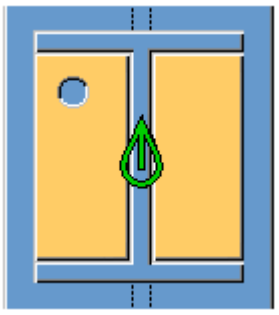
Use the **Holes - end plate** tab to control the galvanizing holes in the end plate.

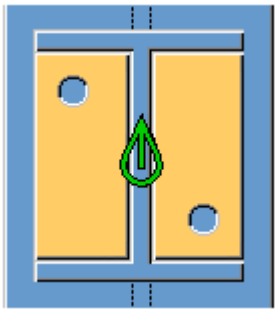
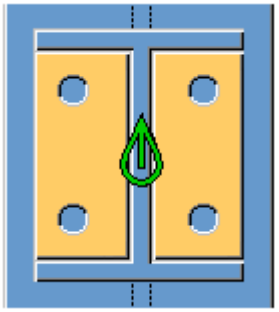
Option	Description
Bolt standard	Select the bolt standard.
Bolt type	Select the bolt type to define the location where the bolts should be attached.
Read data from	You can select to use the `sinkholes.dat` definition file to specify the default values for horizontal and vertical offsets, and the diameters for upper and lower holes. The file is searched in the following order: Environment common system

Option	Description
	steel folder (. .\Environments\common\system\Steel), model folder, XS_FIRM, XS_PROJECT and XS_SYSTEM folder. You can also select to define the holes in the component dialog box.

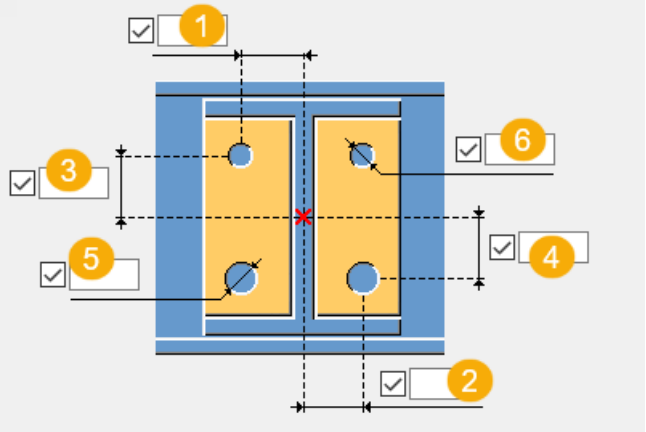
Number of holes

The center of a hole group is the middle point of the beam and the middle point of the haunch, if the haunch exists. The hole groups are composed of 0, 1, 2 or 4 holes.

Option	Description
	Default No holes AutoDefaults can change this option.
	No holes
	1 hole

Option	Description
	2 holes
	4 holes

Hole positions



	Description
1	Horizontal distance between the secondary beam center and the upper hole.
2	Horizontal distance between the secondary beam center and the lower hole.
3	Vertical distance between the secondary beam center and the upper hole.
4	Vertical distance between the secondary beam center and the lower hole.
5	Diameter of the lower hole.

	Description
6	Diameter of the upper hole.

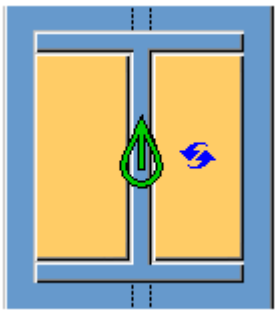
Holes - front plate tab

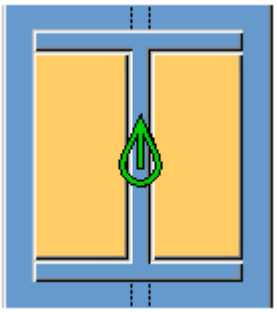
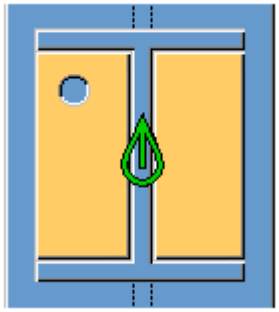
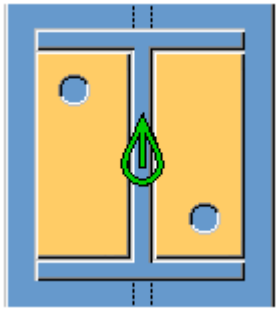
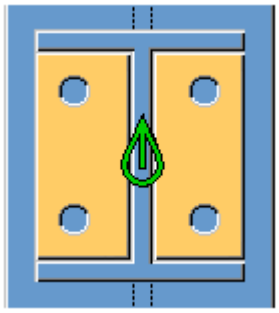
Use the **Holes - front plate** tab to control the galvanizing holes in the front plate.

Option	Description
Bolt standard	Select the bolt standard.
Bolt type	Select the bolt type to define the location where the bolts should be attached.
Read data from	You can select to use the `sinkholes.dat` definition file to specify the default values for horizontal and vertical offsets, and the diameters for upper and lower holes. The file is searched in the following order: Environment common system steel folder (`..\Environments\common\system\Steel`), model folder, `XS_FIRM`, `XS_PROJECT` and `XS_SYSTEM` folder. You can also select to define the holes in the component dialog box.

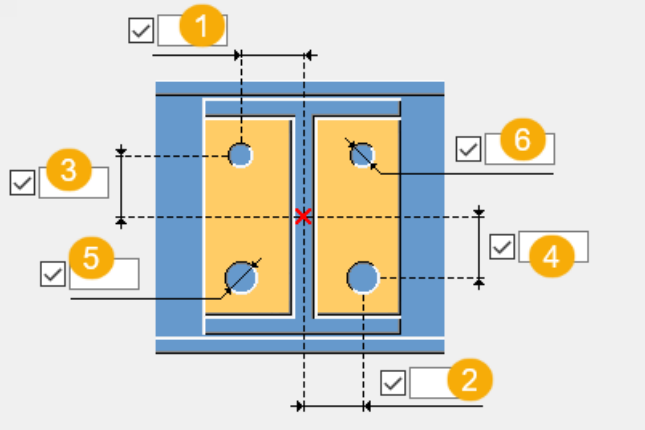
Number of holes

The center of a hole group is the middle point of the beam and the middle point of the haunch, if the haunch exists. The hole groups are composed of 0, 1, 2 or 4 holes.

Option	Description
	Default No holes AutoDefaults can change this option.

Option	Description
	No holes
	1 hole
	2 holes
	4 holes

Hole positions



	Description
1	Horizontal distance between the secondary beam center and the upper hole.
2	Horizontal distance between the secondary beam center and the lower hole.
3	Vertical distance between the secondary beam center and the upper hole.
4	Vertical distance between the secondary beam center and the lower hole.
5	Diameter of the lower hole.
6	Diameter of the upper hole.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Dstv connection properties

Click the link below to find out more:

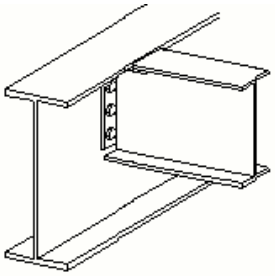
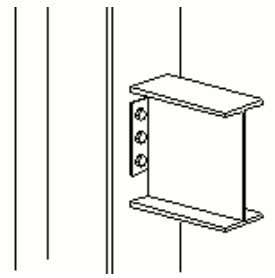
End plate (101)

End plate (101) connects a beam to another beam or to a column using an end plate. The end plate is welded to the secondary beam and bolted to the main part (beam or column).

Objects created

- End plate
- Bolts
- Welds
- Cuts

Use for

Situation	Description
	Beam to beam connection with a bolted end plate.
	Beam to column connection with a bolted end plate.

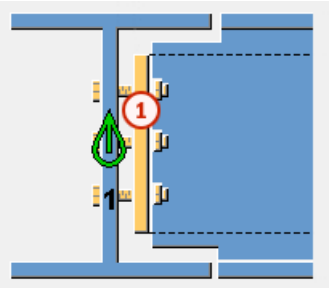
Selection order

1. Select the main part (beam or column).

2. Select the secondary part (beam).

   The connection is created automatically when the secondary beam is selected.

Part identification key

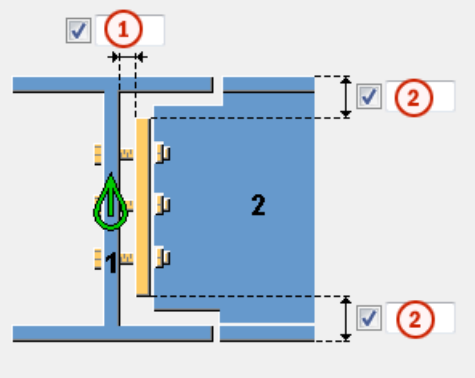


	Part
1	End plate

Picture tab

Use the **Picture** tab to control the position of the end plate.

End plate position



	Description	Default
1	Gap between the main part and the end plate.	2 mm
2	End plate edge distance to the secondary part flange.	

End plate tab

Use the **End plate** tab to control the properties of the end plate.

Plate

Part	Description
End plate	Define the end plate thickness and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Option	Description
End plate edge type	Define how the end plate is cut. The default is **Rolled/Sawn**.
Adapt to secondary part slope	Select whether to align the end plate with the secondary part slope.

Notch tab

Use the **Notch** tab to create notches for the secondary beam and to control the notch properties.

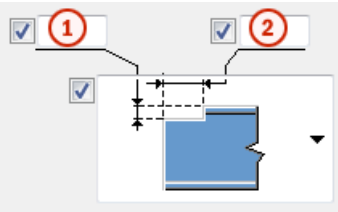
BCSA notch definition

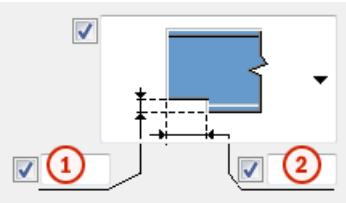
Define whether the notch is created according to British Constructional Steelwork Association (BCSA) specifications.

Option	Description
Default	Notch dimensions.
Yes	Creates a 50 mm notch for simple beam-to-beam connections.
No	Use the options on this **Notch** tab to define the notch dimensions.

Notch dimensions

Define the top and the bottom dimensions of the notch if you have set the **BCSA notch def** option to **No**.

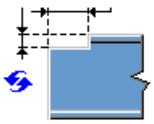
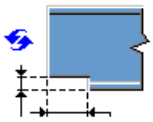
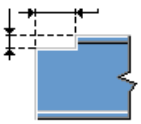
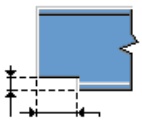


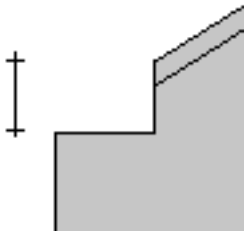
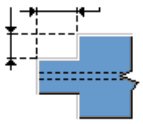
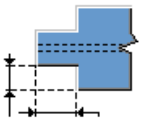
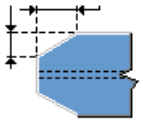
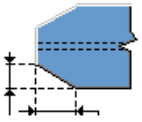
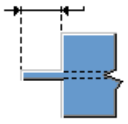
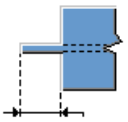
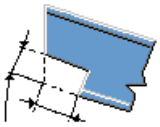


	Description
1	Vertical notch dimension.
2	Horizontal notch dimension.

Notch shape

Define the notch shape for the top and the bottom of the secondary beam.

Option	Option	Description
		Default Creates a square notch on the top side or on the bottom side of the secondary beam. AutoDefaults can change this option.
		No notch
		Creates a square notch on the top side or on the bottom side of the secondary beam. Define the notch dimensions. In beam-to-beam connections with a sloped secondary beam,

Option	Option	Description
		the depth is measured as shown in the picture. 
		Creates a notch on both sides of the secondary part. Define the notch dimensions.
		Creates a chamfered notch on both sides of the secondary beam. Define the chamfer dimensions.
		Creates a strip. Define the length of the strip. The flanges are cut completely.
		Creates a special type of square notch. Define the notch dimensions. The notch is square to the secondary beam. There are no default values for the length or the depth.

Notching side

Define on which side of the secondary beam the notch is created. You can define the side for both the top and the bottom of the secondary beam.

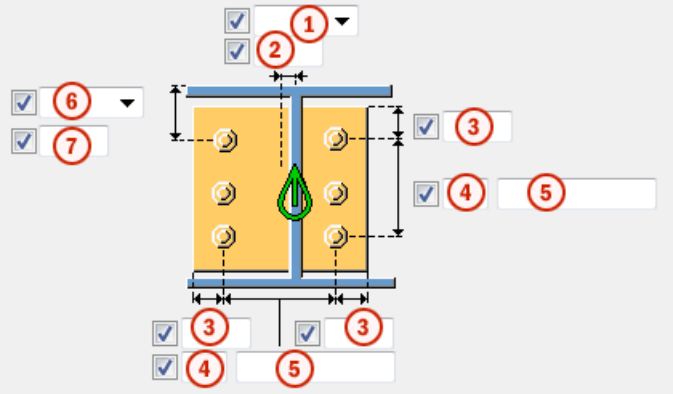
Option	Description
	Default Creates notches on both sides. AutoDefaults can change this option.

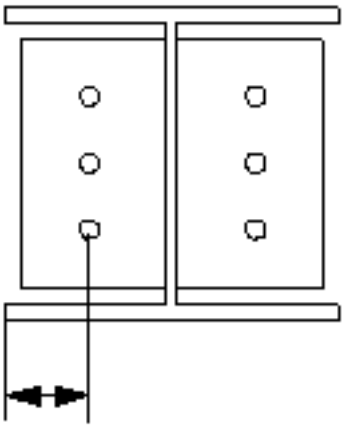
Option	Description
	Creates notches on both sides.
	Creates a notch on the left side.
	Creates a notch on the right side.

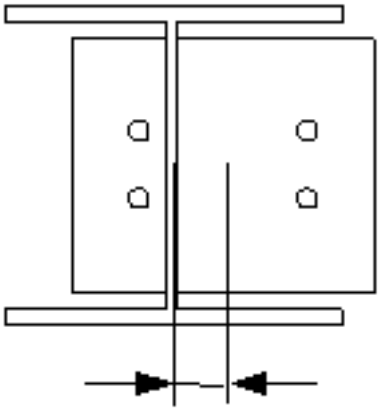
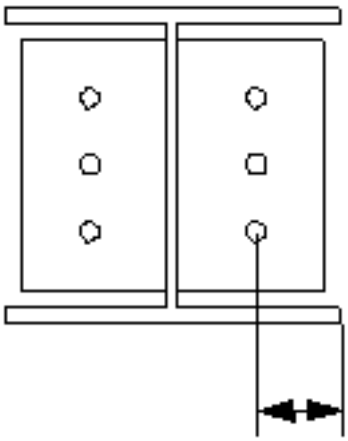
Bolts tab

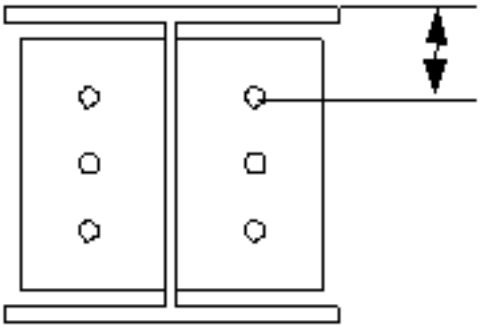
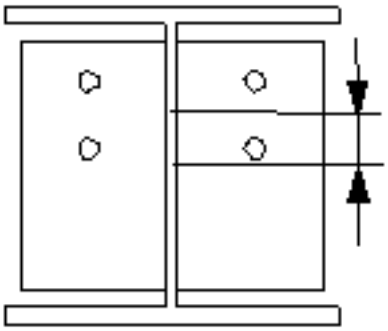
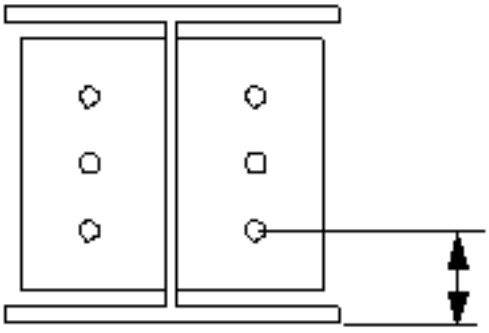
Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt. 

	Description
	• **Middle**: From the center line of the secondary part to the center line of the bolts.  • **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

	Description
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Dimension for vertical bolt group position.

Staggering of bolts

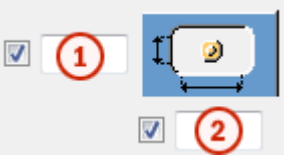
Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

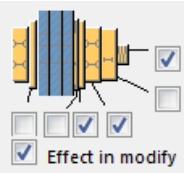


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

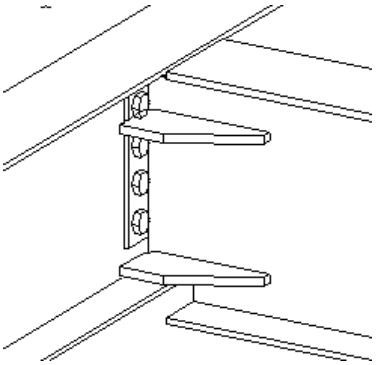
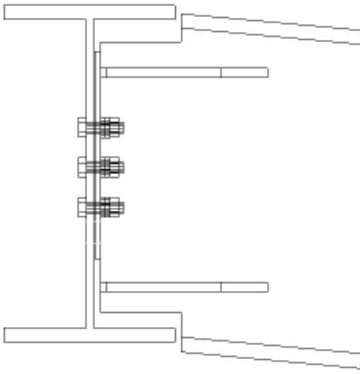
End plate with compensating flange plates (111)

End plate with compensating flange plates (111) connects a beam to another beam using an end plate with compensating flange plates. The end plate is welded to the secondary beam and bolted to the main beam.

Objects created

- End plate
- Compensating flange plates
- Bolts
- Welds
- Cuts

Use for

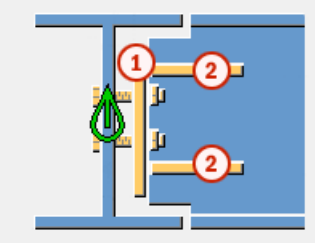
Situation	Description
	Connection with an end plate and compensating flanges.
	Connection with an end plate and compensating flange plates. The secondary beam is sloped.

Selection order

1. Select the main part (beam).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

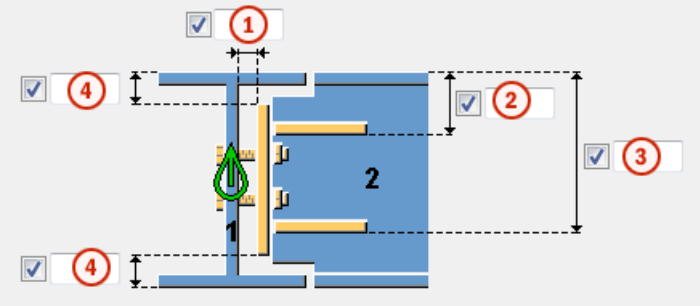


	Part
1	End plate
2	Compensating flange plate

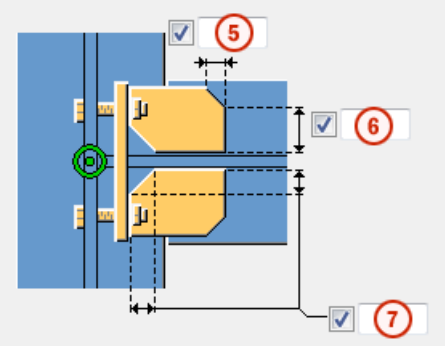
Picture tab

Use the **Picture** tab to control the position of the plates.

Dimensions



	Description
1	Gap between the main beam and the end plate.
2	Distance from the inner edge of the flange plate to the secondary beam flange.
3	Distance from the outer edge of the flange plate to the secondary beam flange.
4	End plate edge distance to the main beam flange.



	Description
1	Horizontal chamfer dimension of the flange plate.
2	Flange plate dimension that remains when a chamfer is created.
3	Inner chamfer dimension of the flange plate.

Parts tab

Use the **Parts** tab to control the plate properties.

Plate

Part	Description
End plate	Define the end plate thickness and height.
Compensating flange plate	Define the flange plate thickness, width, and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Notch tab

Use the **Notch** tab to create notches for the secondary beam and to control the notch properties.

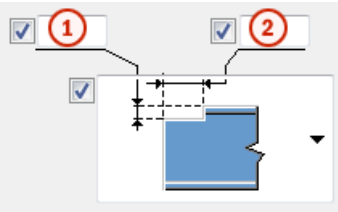
BCSA notch definition

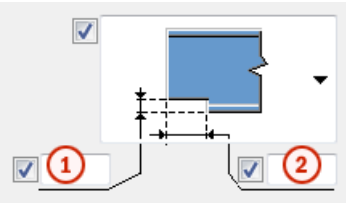
Define whether the notch is created according to British Constructional Steelwork Association (BCSA) specifications.

Option	Description
Default	Notch dimensions.
Yes	Creates a 50 mm notch for simple beam-to-beam connections.
No	Use the options on this **Notch** tab to define the notch dimensions.

Notch dimensions

Define the top and the bottom dimensions of the notch if you have set the **BCSA notch def** option to **No**.

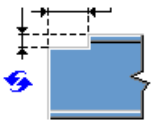
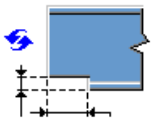
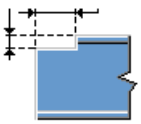
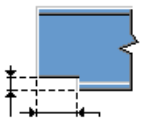


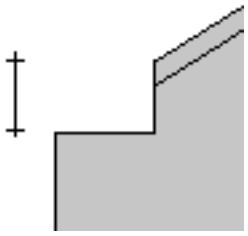
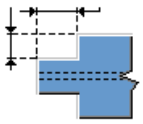
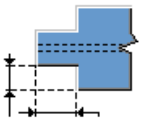
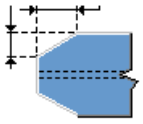
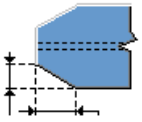
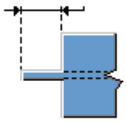
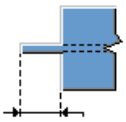
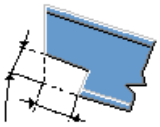


	Description
1	Vertical notch dimension.
2	Horizontal notch dimension.

Notch shape

Define the notch shape for the top and the bottom of the secondary beam.

Option	Option	Description
		Default Creates a square notch on the top side or on the bottom side of the secondary beam. AutoDefaults can change this option.
		No notch
		Creates a square notch on the top side or on the bottom side of the secondary beam. Define the notch dimensions. In beam-to-beam connections with a sloped secondary beam,

Option	Option	Description
		the depth is measured as shown in the picture. 
		Creates a notch on both sides of the secondary part. Define the notch dimensions.
		Creates a chamfered notch on both sides of the secondary beam. Define the chamfer dimensions.
		Creates a strip. Define the length of the strip. The flanges are cut completely.
		Creates a special type of square notch. Define the notch dimensions. The notch is square to the secondary beam. There are no default values for the length or the depth.

Notching side

Define on which side of the secondary beam the notch is created. You can define the side for both the top and the bottom of the secondary beam.

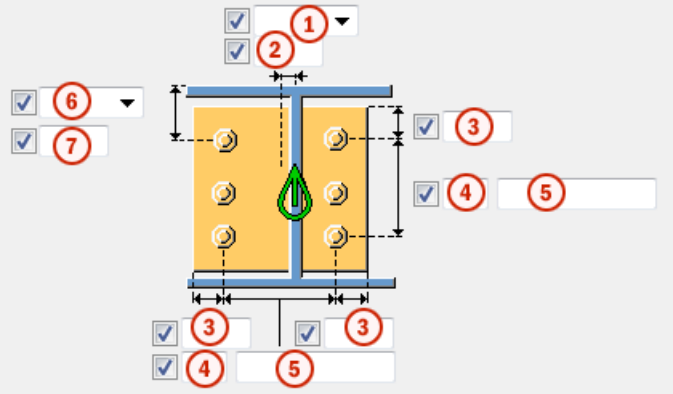
Option	Description
	Default Creates notches on both sides. AutoDefaults can change this option.

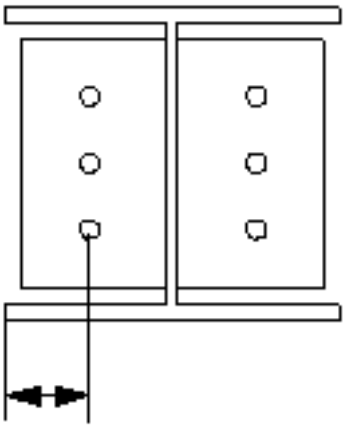
Option	Description
	Creates notches on both sides.
	Creates a notch on the left side.
	Creates a notch on the right side.

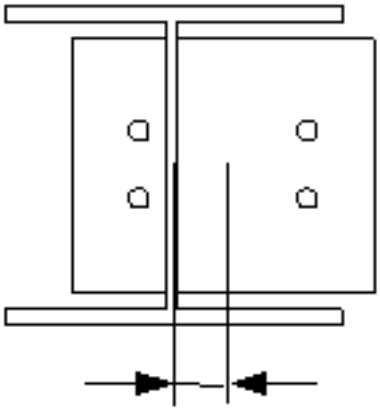
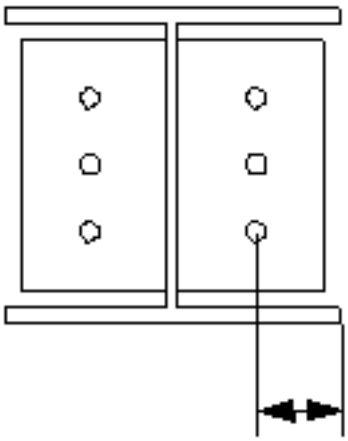
Bolts tab

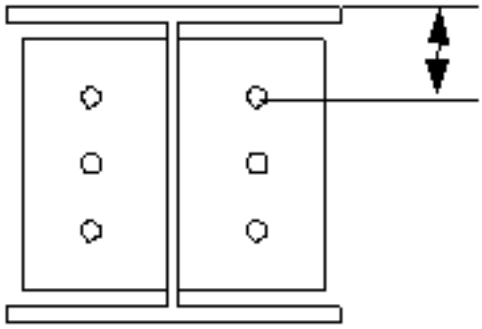
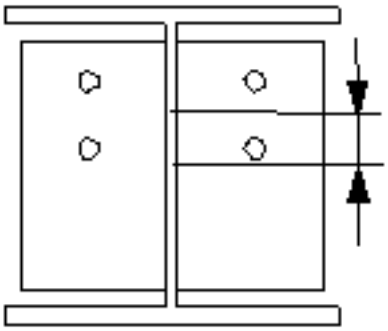
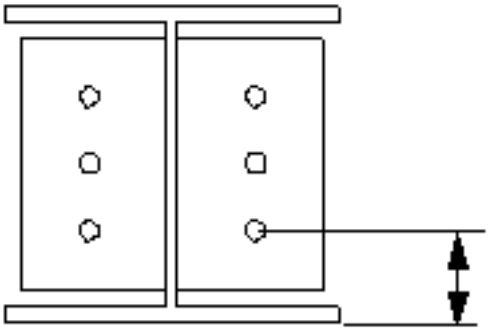
Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt. 

	Description
	• **Middle**: From the center line of the secondary part to the center line of the bolts.  • **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

	Description
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Dimension for vertical bolt group position.

Staggering of bolts

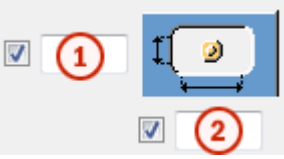
Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

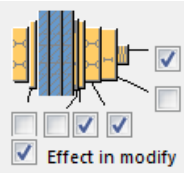


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

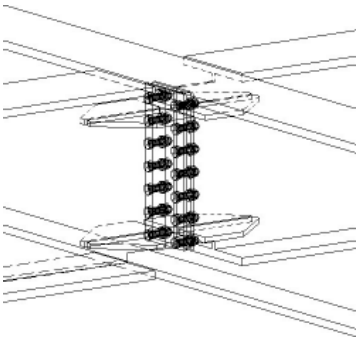
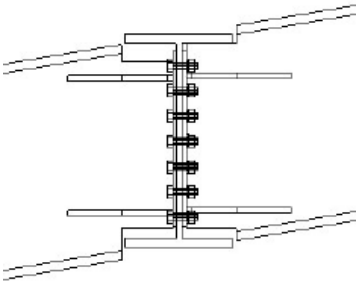
Two-sided end plate with compensating flange plates (112)

Two-sided end plate with compensating flange plates (112) connects a beam to two beams using end plates with compensating flange plates. The end plates are welded to the secondary beams and bolted to the main beam.

Objects created

- End plates
- Compensating flange plates
- Bolts
- Welds
- Cuts

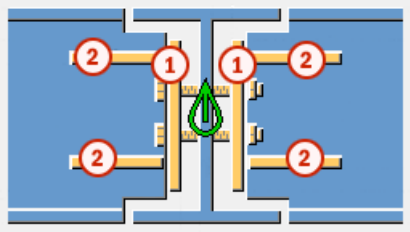
Use for

Situation	Description
	Connection with end plates and compensating flange plates.
	Connection with end plates and compensating flange plates. Secondary beams are sloped.

Selection order

1. Select the main part (beam).
2. Select the first secondary part (beam).
3. Select the second secondary part (beam).
4. Click the middle mouse button to create the connection.

Part identification key

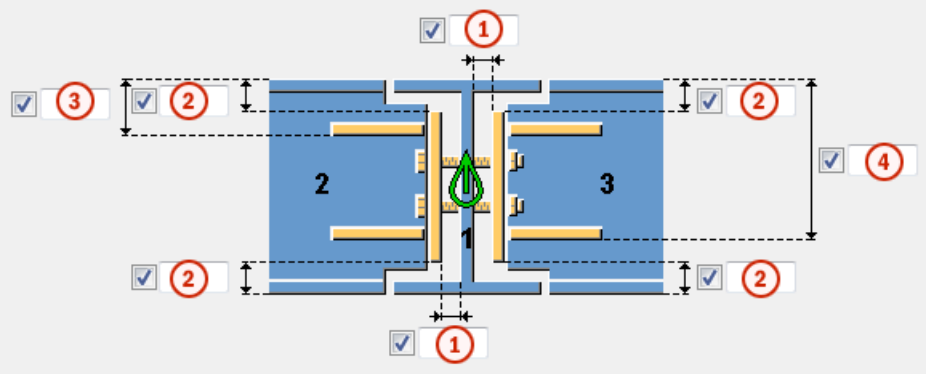


	Part
1	End plate
2	Compensating flange plate

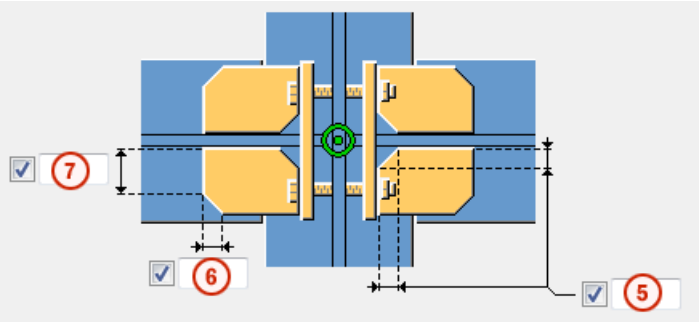
Picture tab

Use the **Picture** tab to control the position of the end plates and compensating flange plates.

Dimensions



	Description
1	Gap between the main beam and the end plate.
2	End plate edge distance to the secondary beam flange.
3	Distance from the inner edge of the flange plate to the secondary beam flange.
4	Distance from the outer edge of the flange plate to the secondary beam flange.



	Description
1	Inner chamfer dimension of the flange plate.
2	Horizontal chamfer dimension of the flange plate.
3	Flange plate dimension that remains when a chamfer is created.

Parts tab

Use the **Parts** tab to control the plate properties.

Plate

Part	Description	Default
End plate, **Second end plate**	Define the end plate thickness and height. If you have defined the gap between the main beam and the end plate on the **Picture** tab, the length entered on the **Parts** tab is not taken into account.	If the secondary part width is less than `200mm`, the thickness of the end plate is `8mm`. Otherwise, it is `10mm`.
Compensating flange plate, **Second compensating flange plate**	Define the flange plate thickness, width, and height. The same values are used for the top and the bottom plates.	**Width** = The default value is based on rounding (beam flange width - beam web thickness) / 2.0. **Height** = `150mm` more than the notch length

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Move plates 2 in y-direction

You can move the flange plates and the end plate on the second secondary beam side in the y-direction. By default, the plates on the second secondary side are positioned so that the holes are symmetric. To use this option, set the horizontal bolt group position to the **Middle** position and define the horizontal bolt group dimension on the **Bolts** tab. Moving the plates is useful especially when the secondary beams are skewed or curved.

Notch tab

Use the **Notch** tab to create notches for the secondary beams and to control the notch properties. Define the notches for both secondary beams.

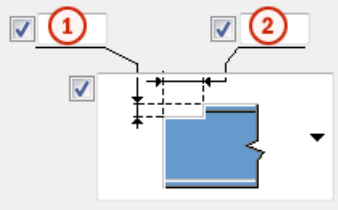
BCSA notch definition

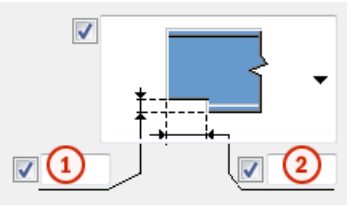
Define whether the notch is created according to British Constructional Steelwork Association (BCSA) specifications.

Option	Description
Default	Notch dimensions.
Yes	Creates a 50 mm notch for simple beam-to-beam connections.
No	Use the options on this **Notch** tab to define the notch dimensions.

Notch dimensions

Define the top and the bottom dimensions of the notch if you have set the **BCSA notch def** option to **No**.

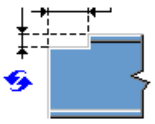
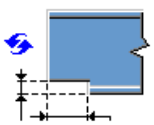


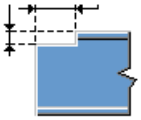
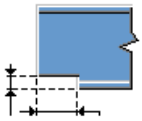
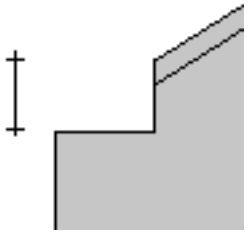
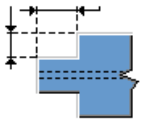
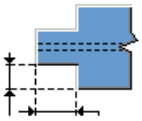
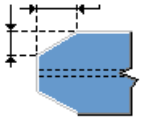
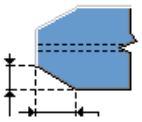
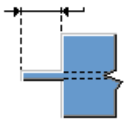
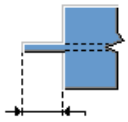
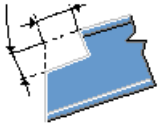
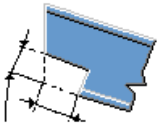


	Description
1	Vertical notch dimension.
2	Horizontal notch dimension.

Notch shape

Define the notch shape for the top and the bottom of the secondary beam.

Option	Option	Description
		Default Creates a square notch on the top side or on the

Option	Option	Description
		bottom side of the secondary beam. AutoDefaults can change this option.
		No notch
		Creates a square notch on the top side or on the bottom side of the secondary beam. Define the notch dimensions. In beam-to-beam connections with a sloped secondary beam, the depth is measured as shown in the picture. 
		Creates a notch on both sides of the secondary part. Define the notch dimensions.
		Creates a chamfered notch on both sides of the secondary beam. Define the chamfer dimensions.
		Creates a strip. Define the length of the strip. The flanges are cut completely.
		Creates a special type of square notch. Define the notch dimensions. The notch is

Option	Option	Description
		square to the secondary beam. There are no default values for the length or the depth.

Notching side

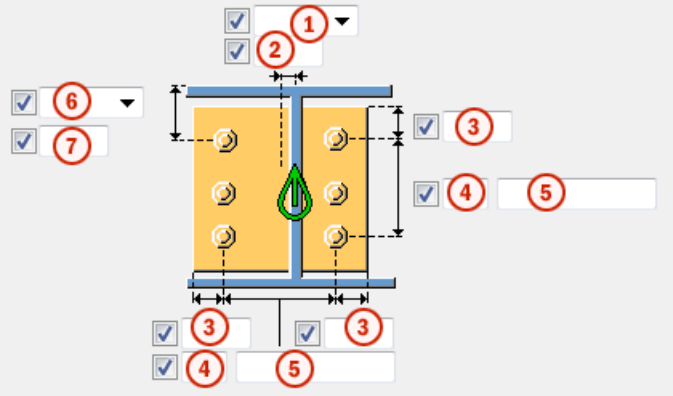
Define on which side of the secondary beam the notch is created. You can define the side for both the top and the bottom of the secondary beam.

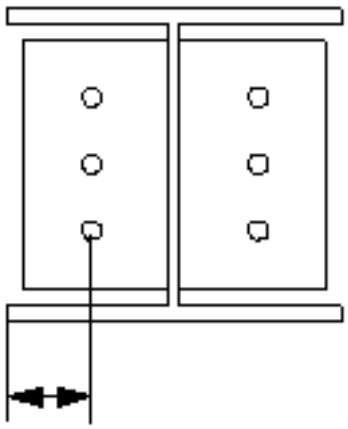
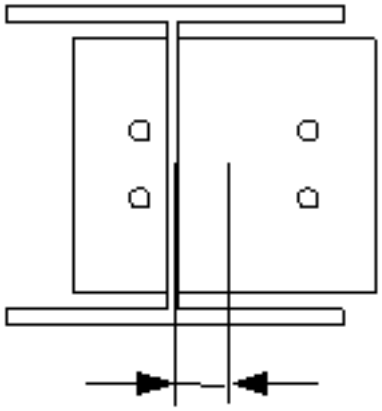
Option	Description
	Default Creates notches on both sides. AutoDefaults can change this option.
	Creates notches on both sides.
	Creates a notch on the left side.
	Creates a notch on the right side.

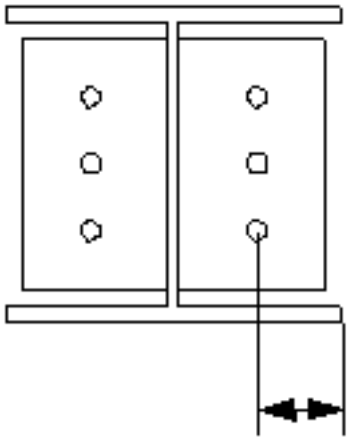
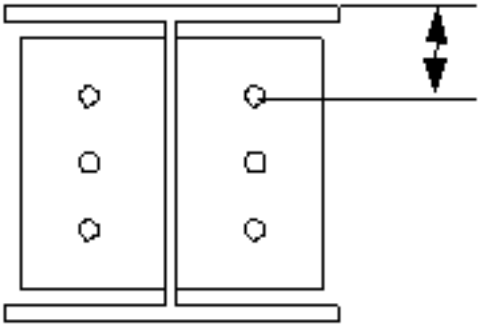
Bolts tab

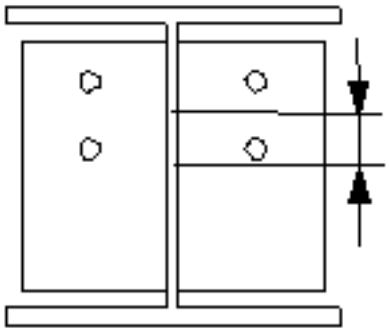
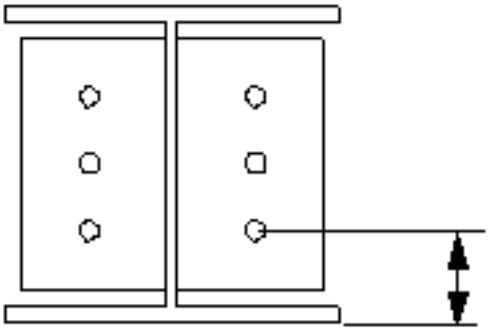
Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt.  • **Middle**: From the center line of the secondary part to the center line of the bolts. 

	Description
	• **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Dimension for vertical bolt group position.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2

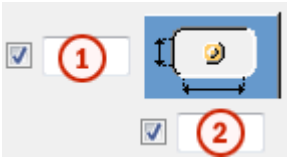
Option	Description
	Staggered type 3
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



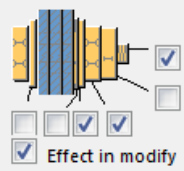
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

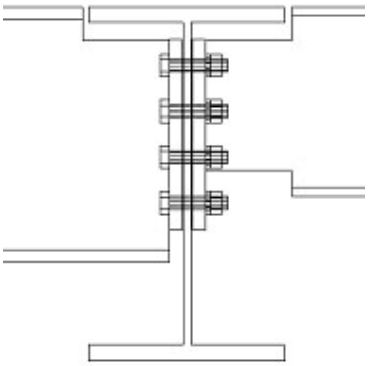
Two sided end plate (115)

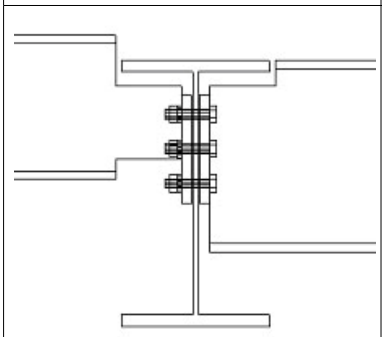
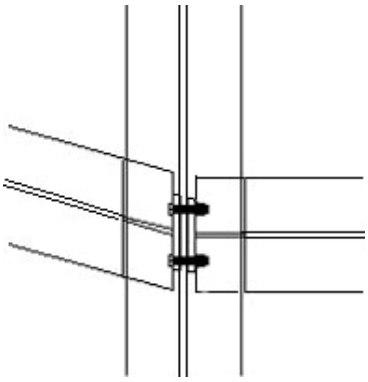
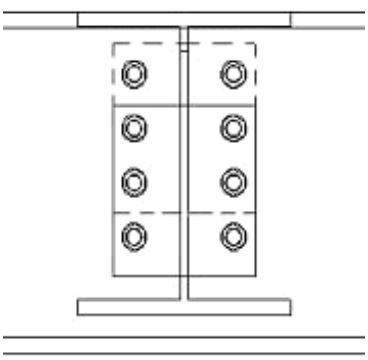
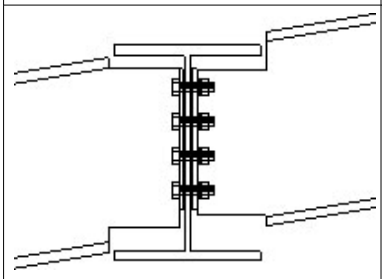
Two sided end plate (115) connects two beams to a beam or a column using end plates. The end plates are welded to the secondary beams and bolted to the main part (beam or column).

Objects created

- End plates (2)
- Bolts
- Welds
- Cuts

Use for

Situation	Description
	End plates with two secondary beams. Automatic notching for bolt clearance.

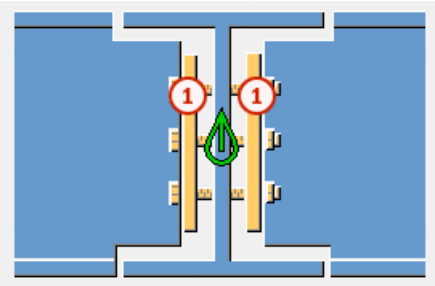
Situation	Description
	End plates with two secondary beams at different heights.
	End plates with two secondary beams. A square and a skewed secondary beam.
	End plates with two secondary beams. Safety connection.
	End plates with two sloped secondary beams. Various notching options.

Selection order

1. Select the main part (beam or column).
2. Select the first secondary part (beam).
3. Select the second secondary part (beam).

4. Click the middle mouse button to create the connection.

Part identification key

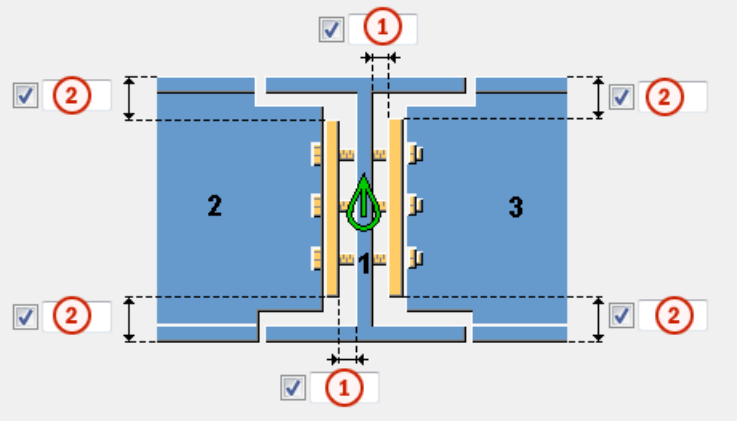


	Part
1	End plate

Picture tab

Use the **Picture** tab to control the position of the end plates.

Dimensions



	Description	Default
1	Gap between the main part and the end plate.	2 mm
2	End plate edge distance to the secondary beam flange.	50 mm

End plates tab

Use the **End plates** tab to control the properties of the end plates.

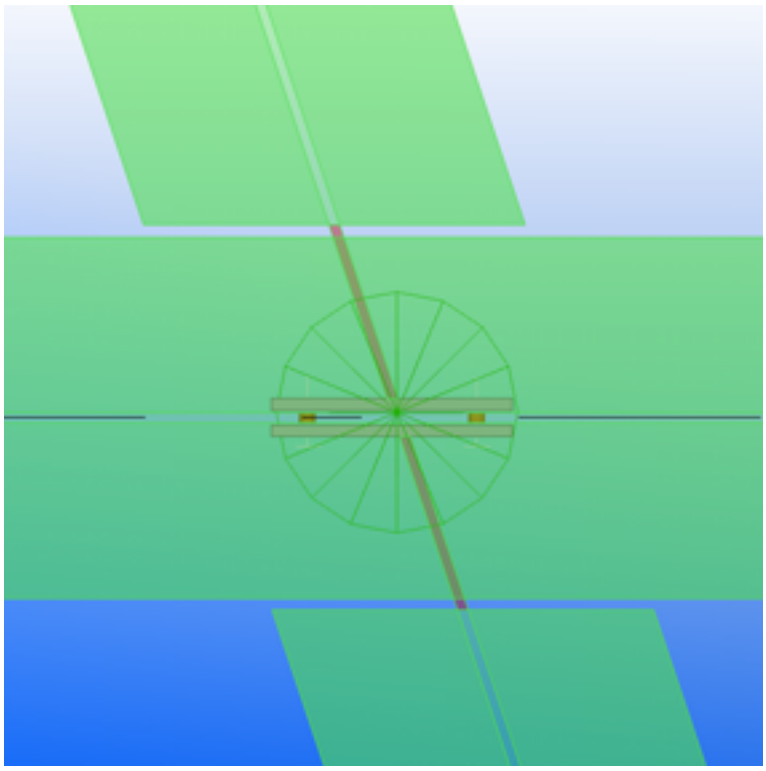
Plate

Part	Description
End plate, **2nd End plate**	Define the end plate thickness and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Move plates 2 in y-direction

You can move the end plate on the second secondary beam side in the y-direction. By default, the plates on the second secondary side are positioned so that the holes are symmetric. To use this option, set the horizontal bolt group position to the **Middle** position and define the horizontal bolt group dimension on the **Bolts** tab. Moving the plates is useful especially when the secondary beams are skewed or curved.



End plate Edge Type

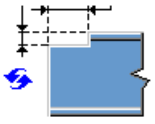
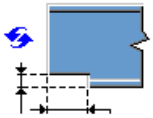
Define how the end plate is cut. The default is **Rolled/Sawn**.

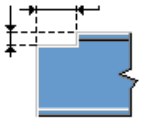
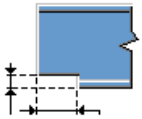
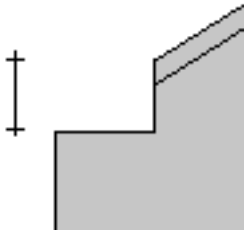
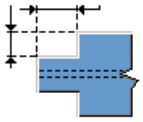
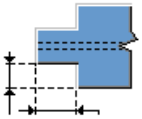
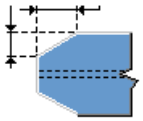
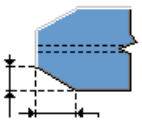
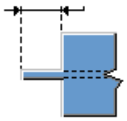
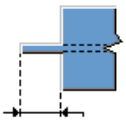
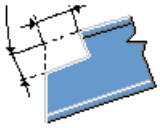
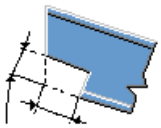
Notch tab

Use the **Notch** tab to create notches for the secondary beams and to control the notch properties. Define the notches for both secondary beams.

Notch shape

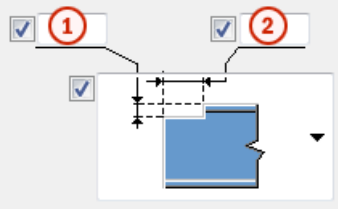
Define the notch shape for the top and the bottom of the secondary beam.

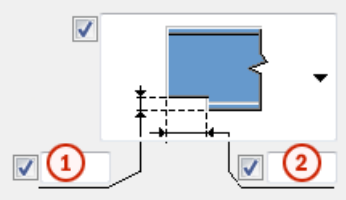
Option	Option	Description
		Default Creates a square notch on the top side or on the bottom side of the secondary beam. AutoDefaults can change this option.
		No notch

Option	Option	Description
		Creates a square notch on the top side or on the bottom side of the secondary beam. Define the notch dimensions. In beam-to-beam connections with a sloped secondary beam, the depth is measured as shown in the picture. 
		Creates a notch on both sides of the secondary part. Define the notch dimensions.
		Creates a chamfered notch on both sides of the secondary beam. Define the chamfer dimensions.
		Creates a strip. Define the length of the strip. The flanges are cut completely.
		Creates a special type of square notch. Define the notch dimensions. The notch is square to the secondary beam. There are no default values for the length or the depth.

Notch dimensions

Define the top and the bottom dimensions of the notch if you have set the **BCSA notch def** option to **No**.





	Description
1	Vertical notch dimension.
2	Horizontal notch dimension.

Notching side

Define on which side of the secondary beam the notch is created. You can define the side for both the top and the bottom of the secondary beam.

Option	Description
	Default Creates notches on both sides. AutoDefaults can change this option.
	Creates notches on both sides.
	Creates a notch on the left side.
	Creates a notch on the right side.

BCSA notch definition

Define whether the notch is created according to British Constructional Steelwork Association (BCSA) specifications.

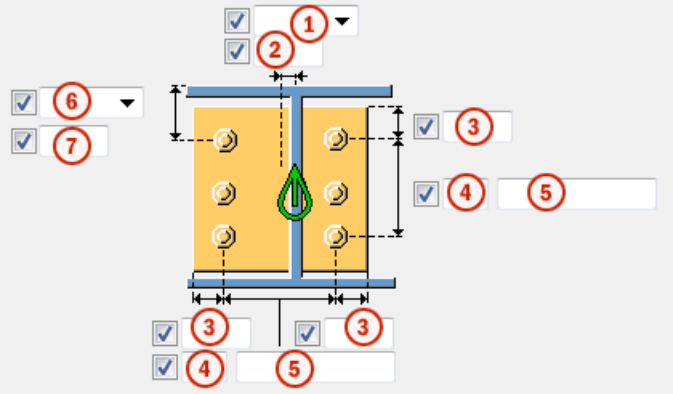
Option	Description
Default	Notch dimensions.
Yes	Creates a 50 mm notch for simple beam-to-beam connections.

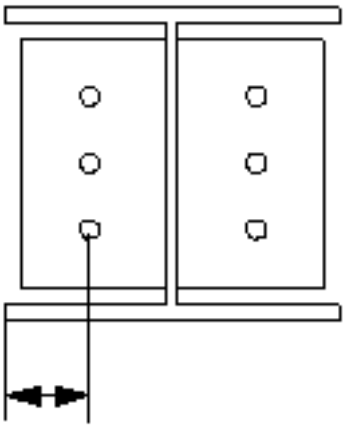
Option	Description
No	Use the options on this **Notch** tab to define the notch dimensions.

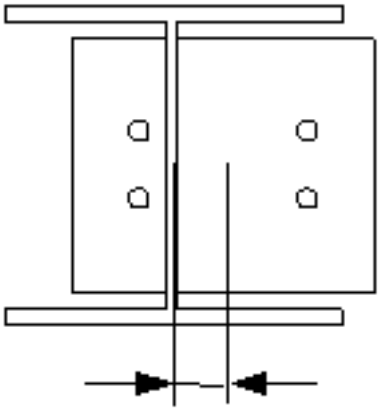
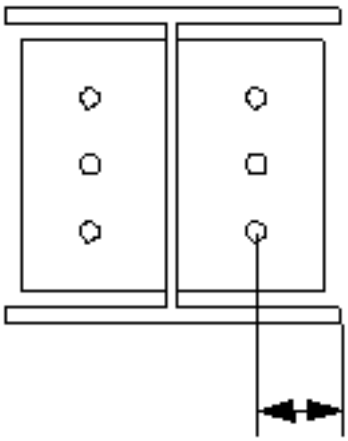
Bolts tab

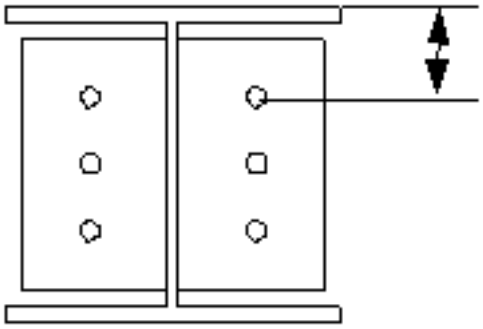
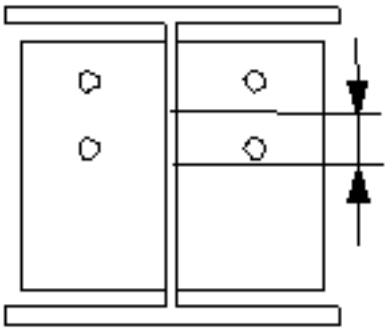
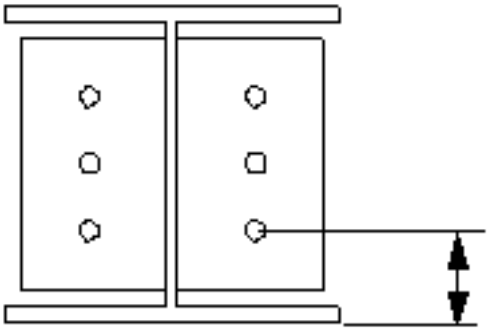
Use the **Bolts** tab to control the bolts properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt. 

	Description
	• **Middle**: From the center line of the secondary part to the center line of the bolts.  • **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

	Description
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Dimension for vertical bolt group position.

Staggering of bolts

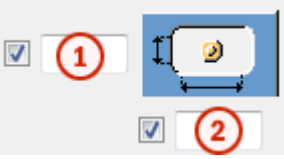
Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

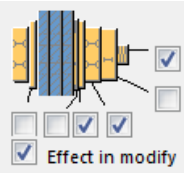


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

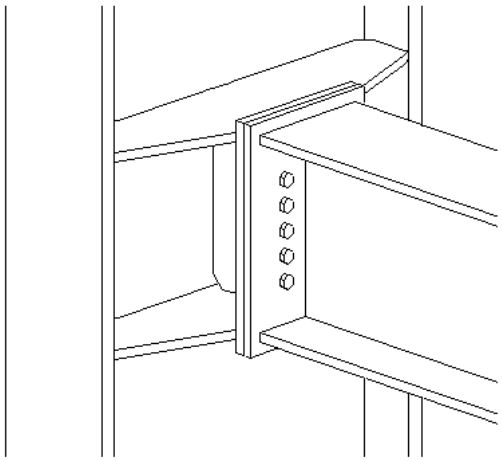
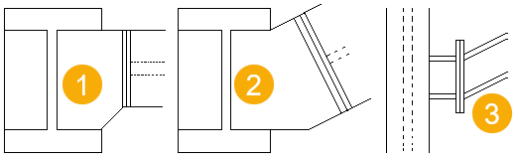
Stub connection (119)

Stub connection (119) creates a connection between an H(I) type of column and a beam. The beam can be of any type. The connection can create eight stiffener plates and four haunch plates. The end plates are bolted to each other, and the other plates are welded.

Objects created

- End plates
- Haunch plates
- Stiffeners
- Bolts
- Welds

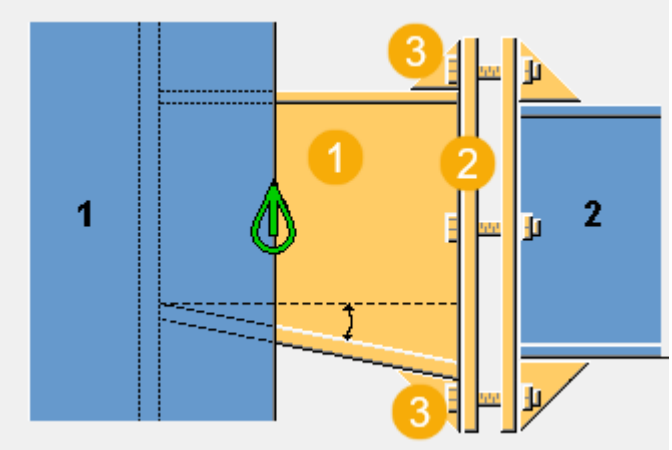
Use for

Situation	Description
	Stub connection with bolted end plates and haunch plates.
	The secondary part can be skewed or sloped. Top and bottom haunch plates can be created. **1** Secondary part web not directed to the center of the column (top view). **2** Secondary part web not perpendicular to the column horizontally (top view). **3** Secondary part web not perpendicular to the column vertically (side view).

Selection order

1. Select the main part (column).
2. Select the secondary part (beam).

Part identification key

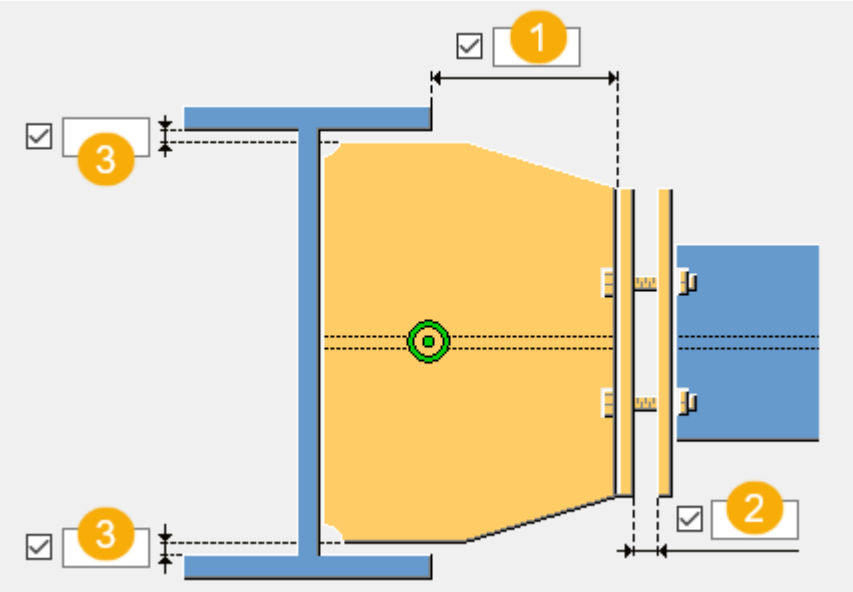


	Description
1	Stiffener plates
2	End plates
3	Haunch plates

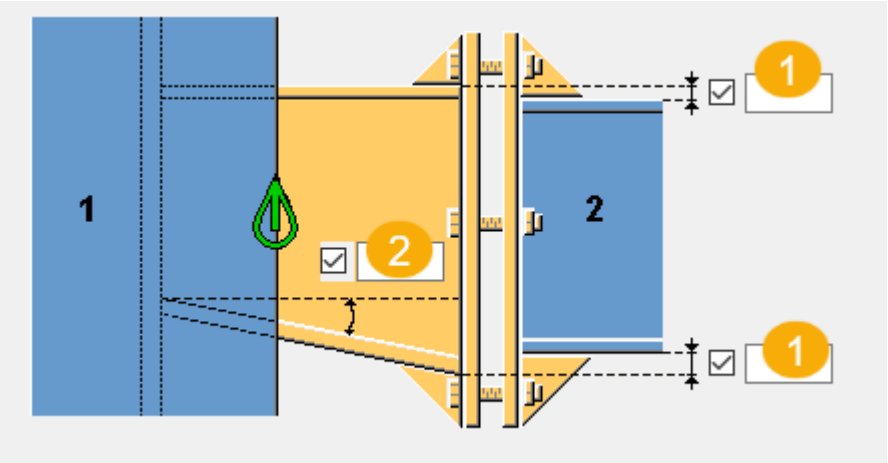
Picture tab

Use the **Picture** tab to define end plate and stiffener dimensions.

Dimensions



	Description	Default
1	Distance from the column to the first end plate. If the secondary part web is not perpendicular to the column horizontally, this is the minimum distance from the column to the first end plate.	100 mm
2	Distance between the end plates.	0 mm
3	Gap between the end plate and the beam web.	



	Descriptioin	Default
1	Distance between the exterior face of the beam and the exterior face of the front stiffener.	0 mm
2	Angle between the bottom front stiffener and the horizon. (This angle is valid only in the regular case).	0 degrees

Parts tab

Use the **Parts** tab to define the part properties.

Parts

Option	Description
Top plate **Bottom plate** **Middle plate**	Thickness of the top, bottom and middle plate.
Upper haunches **Lower haunches**	Thickness of the upper and lower haunches.
End plate 1	Thickness, width and height of the end plate.

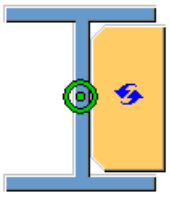
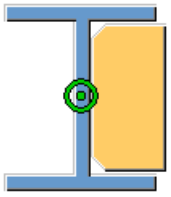
Option	Description
End plate 2	Thickness of the end plate.
Top stiffener **Bottom stiffener** **Middle stiffener**	Thickness of the top, bottom and middle stiffener.

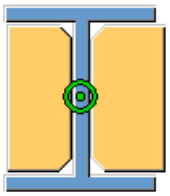
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

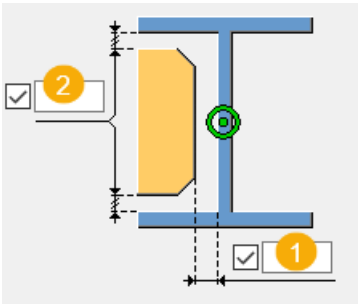
Use the **Parameters** tab to control the stiffener creation, position and chamfers.

Stiffener creation

Option	Description
	Default Full Creates a full stiffener of the same height as the web of the main part. AutoDefaults can change this option.
	Full Creates a full stiffener of the same height as the web of the main part.

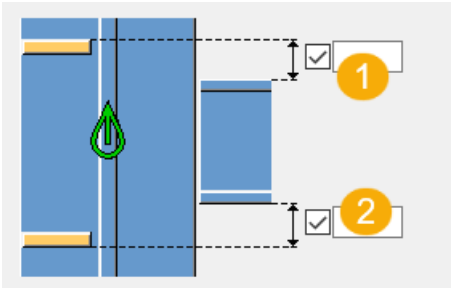
Option	Description
	Full, both sides of the main part Creates a full stiffener of the same height as the web of the main part.

Stiffener gap



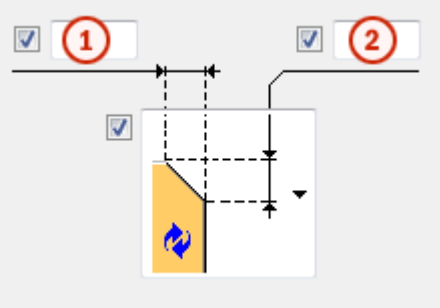
	Description
1	Distance between the stiffener and the column web
2	Distance between the stiffener and the column flange

Stiffener positions



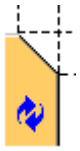
	Description
1	Size of the gap between the top stiffener and the beam flange edge.
2	Size of the gap between the bottom stiffener and the beam flange edge.

Chamfer dimensions



	Description
1	Horizontal dimension of the chamfer.
2	Vertical dimension of the chamfer.

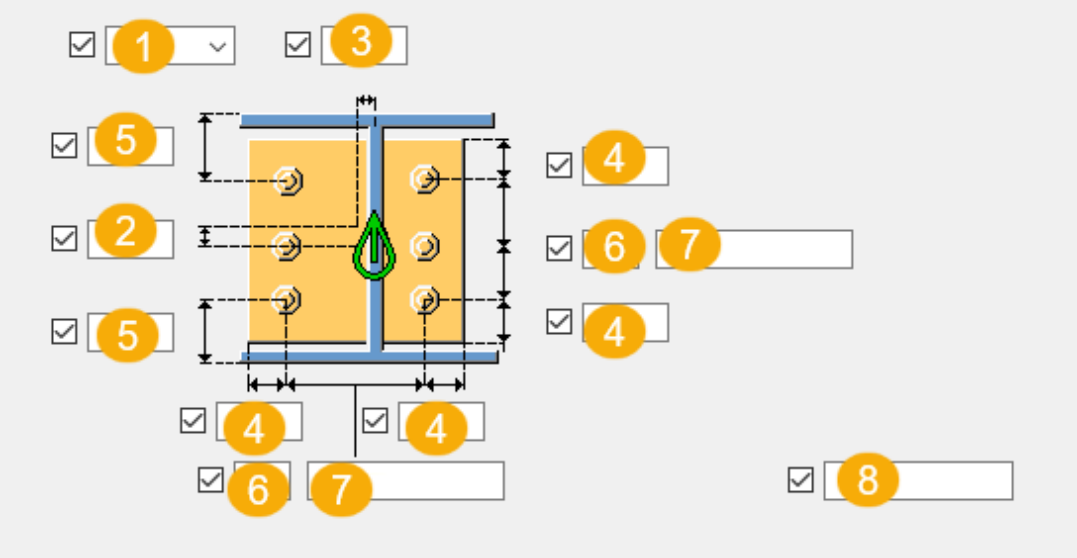
Chamfer type

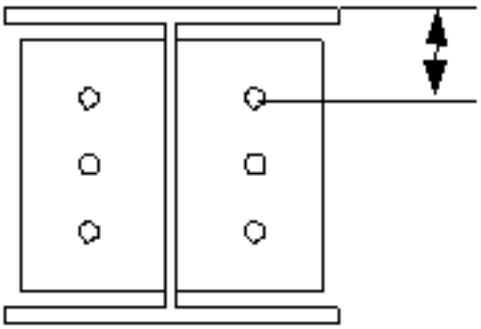
Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

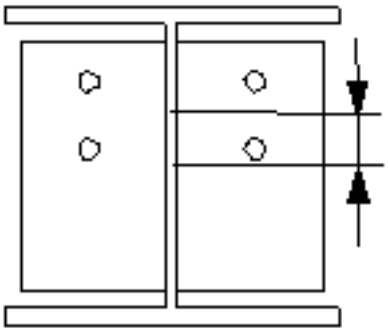
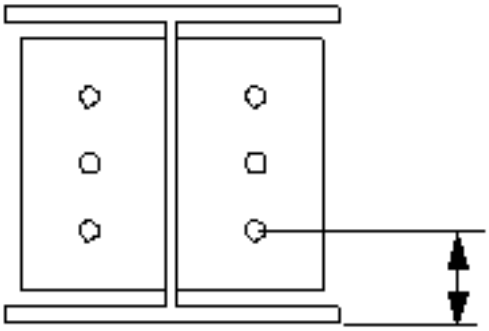
Bolts tab

Use the **Bolts** tab to define the bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

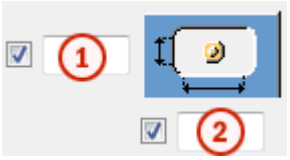
	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
2	Dimension for vertical bolt group position.
3	Dimension for horizontal bolt group position.
4	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
5	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
6	Number of bolts.
7	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
8	Define which bolts are deleted from the bolt group.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



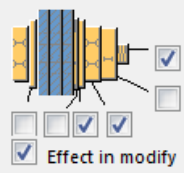
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	

Option	Description	Default
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3

Option	Description
	Staggered type 4

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

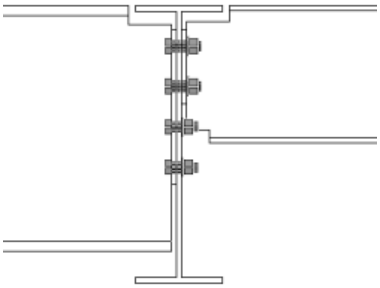
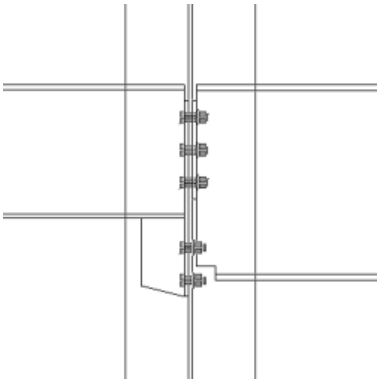
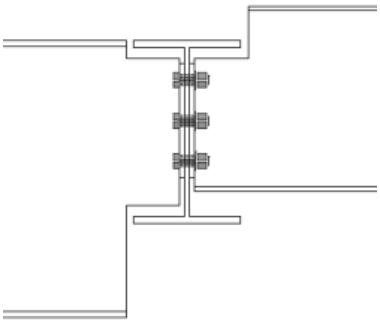
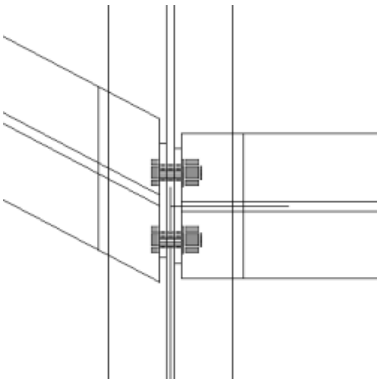
Two sided end plate (142)

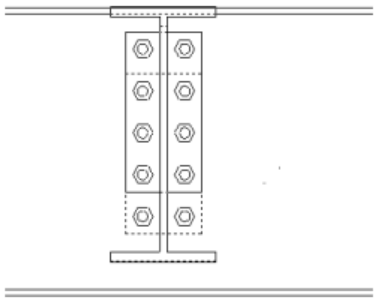
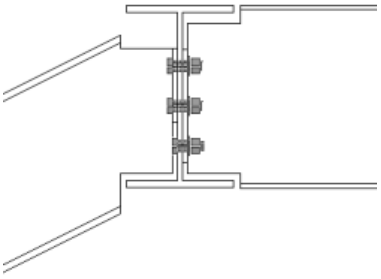
Two sided end plate (142) connects two beams to a beam or to a column using bolted end plates. One bolt group goes through all the three parts.

Objects created

- End plates
- Shim plates
- Compensating flange plates (optional)
- Haunch plates (optional)
- Holes
- Bolts
- Welds
- Cuts

Use for

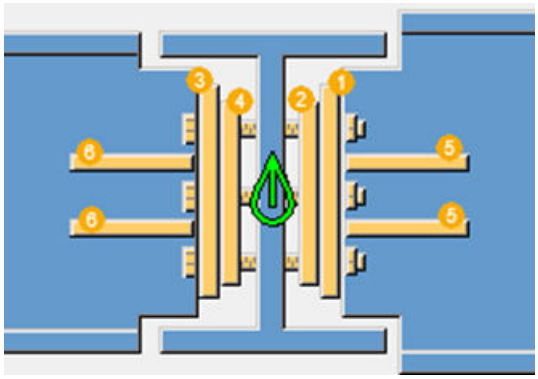
Situation	Description
	End plate connection with two secondary parts. Automatic notching for bolt clearance.
	End plate connection with two secondary parts and with a haunch plate. Automatic notching for bolt clearance.
	End plate connection with two secondary parts at different heights.
	End plate connection with two secondary parts. The secondary parts can be square and/or skewed.

Situation	Description
	End plate connection with two secondary parts. Safety connection.
	End plate connection with two secondary parts. The secondary parts can be leveled and/or sloped.

Selection order

1. Select the main part (column or beam).
2. Select the first secondary part (beam).
3. Select the second secondary part (beam).
4. Click the middle mouse button to create the connection.

Part identification key



	Part
1	End plate for the first secondary part

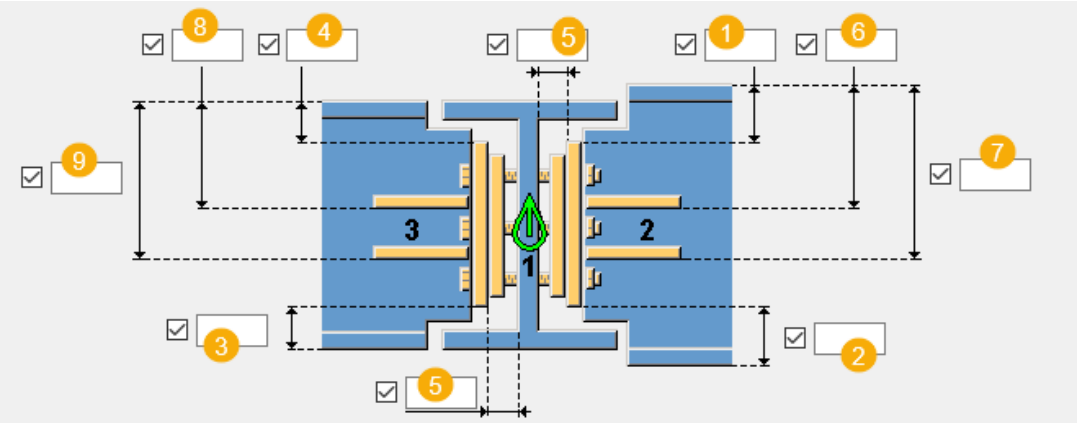
	Part
2	Shim plate for the first secondary part
3	End plate for the second secondary part
4	Shim plate for the second secondary part
5	Compensating flange plate for the first secondary part
6	Compensating flange plate for the second secondary part

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to control the positions of the end plates and compensating flange plates.

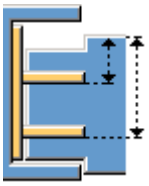
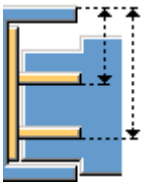
Plate positions



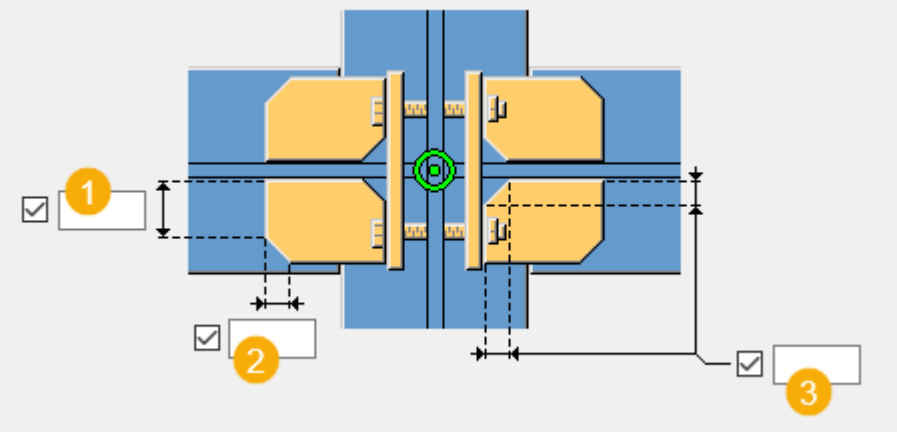
	Description
1	End plate upper edge distance from the top of the first secondary beam.
2	End plate lower edge distance from the bottom of the first secondary beam.
3	End plate lower edge distance from the bottom of the second secondary beam.
4	End plate upper edge distance from the top of the second secondary beam.
5	Gap between the shim plates and the main part. Gap for each side individually. If the shim plates are not used, the defined gap is created between the end plate and the main part.
6	Upper compensation flange plate edge distance from the top of the first secondary beam.

	Description
7	Lower compensation flange plate edge distance from the top of the first secondary beam.
8	Upper compensation flange plate edge distance from the top of the second secondary beam.
9	Lower compensation flange plate edge distance from the top of the first secondary beam.

Compensation flange plate arrangements

Option	Description
	Default Compensation flange plates are not created. AutoDefaults can change this option.
	Compensation flange plates are not created.
	Compensation flange plates are created. Egde distance from the top of the secondary part.
	Compensation flange plates are created. Edge distance from the top of the main part.

Compensating flange plate shapes



	Description
1	Compensating flange plate dimension that remains when a chamfer is created.
2	Horizontal chamfer dimension of the compensating flange plate.
3	Inner chamfer dimension of the compensating flange plate.

Sort secondaries by profile height

When you create a **Two sided end plate (142)** connection, the larger of the two profiles is usually selected as the first secondary beam. If the profile is later changed and the second secondary beam becomes larger than the first secondary beam, the order of secondary beams can be redefined.

- **Yes** switches the secondary beams so that the largest beam automatically becomes the first secondary beam.
- **No** does not change the order of secondary beams if the profile size is changed.

Plates 1 tab

Use the **Plates 1** tab to control the size of the end plate, shim plates and compensating flange plates for the first secondary beam.

Plates

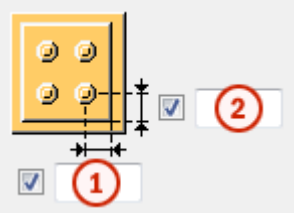
Option	Description	Default
End plate	End plate thickness, width and height.	thickness = `10 mm`

Option	Description	Default
Fitting Plate 1 **Fitting Plate 2** **Fitting Plate 3**	Shim plate thickness. The plate is created only if the plate thickness is given. You can define up to three different shim plates.	0
Number of fitting pl. 1 (DEF=1) **Number of fitting pl. 2 (DEF=1)** **Number of fitting pl. 3 (DEF=1)**	Number of shim plates for each thickness.	By default, 1 plate is created.
Comp. flange pl	Compensation flange plate thickness, width and height.	

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

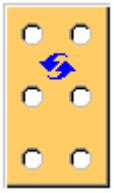
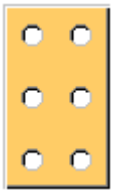
Bolt edge distances in shim plate

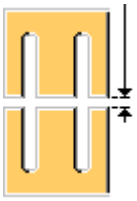
Define the bolt edge distances for shim plates. When these fields are empty, shim plates are of the same size as the end plate.



	Description	Default
1	Horizontal bolt edge distance in the shim plate.	30 mm
2	Vertical bolt edge distance in the shim plate.	30 mm

Shim plate shape

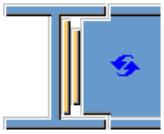
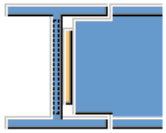
Option	Description
	Default Holes are based on the bolt group of the connection. AutoDefaults can change this option.
	Holes are based on the bolt group of the connection.
	Finger shim plate with horizontal slots. The plate can be installed from the right or the left side of the connection.
	Finger shim plate with vertical slots. The plate can be installed from the top of the connection.

Option	Description
	Two separate finger shim plates with horizontal slots.
	Two separate finger shim plates with vertical slots.

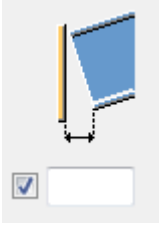
Tolerance

Define the tolerance for the slots in finger shim plates. The width of the slot is the bolt diameter + the tolerance. For two separate shim plates, also define the tolerance between the plates.

Shim plate position

Option	Description
	Default Shim plates are outside the main part. AutoDefaults can change this option.
	Shim plates are outside the main part.
	Shim plates are inside the main part.

Gap size



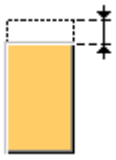
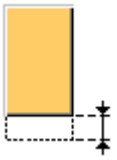
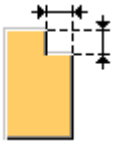
Define the limit value for the gap between the end plate and the secondary beam. Use this when the beam is slightly curved or sloped to decide if the end angle is so small that the beam end can be straight.

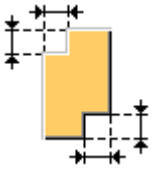
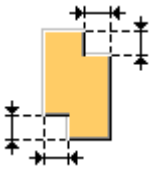
If the actual gap is smaller than this value, the end of the beam is left straight.

If the actual gap is larger than this value, the end of the beam is fitted to the end plate.

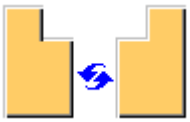
Safety connections

In two-sided connections the safety connection options improve safety during erection. Safety connection options move the end plate or create different notches so that some of the bolts are in single shear instead of double shear. This allows the first secondary beam to be connected while the crane moves to get the next beam.

Option	Description
	Default End plate with no notches. AutoDefaults can change this option.
	End plate with no notches.
	One end plate is moved up to create a safety connection.
	One end plate is moved down to create a safety connection.
	One opposite upper corner of each end plate is notched to create a safety connection
	One opposite upper corner of each end plate is notched to create a safety connection.

Option	Description
	Diagonal opposite corners of each end plate are notched to create a safety connection.
	Diagonal opposite corners of each end plate are notched to create a safety connection.

Safety connection location

Option	Description
	Default Affects both the near side and the far side end plates. AutoDefaults can change this option.
	Affects both the near side and the far side end plates.
	Affects only the near side end plate.
	Affects only the far side end plate.

Safety connection cut type

Option	Description
	Default Square cut. This selection affects only safety connections that are created with cuts. It does not affect safety connections that shorten the plate. AutoDefaults can change this option.

Option	Description
	Square cut
	Line cut
	Concave arc cut

Safety connection cut dimensions

Option	Description
Vertical cut/offset	Define the height of the notch or the vertical offset of the end plate.
Horizontal cut	Define the width of the notch in the end plate.
Radius	Define the radius of the concave arc cut.

Plates 2 tab

Use the **Plates 2** tab to control the size of the end plate, shim plates and compensating flange plates for the second secondary beam.

Plates

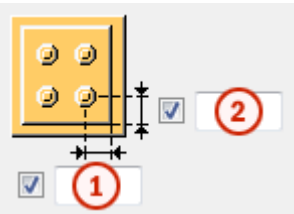
Option	Description	Default
End plate	End plate thickness, width and height.	thickness = `10 mm`
Fitting Plate 1 **Fitting Plate 2** **Fitting Plate 3**	Shim plate thickness. The plate is created only if the plate thickness is given. You can define up to three different shim plates.	`0`

Option	Description	Default
Number of fitting pl. 1 (DEF=1) **Number of fitting pl. 2 (DEF=1)** **Number of fitting pl. 3 (DEF=1)**	Number of shim plates for each thickness.	By default, 1 plate is created.
Comp. flange pl	Compensation flange plate thickness, width and height.	

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

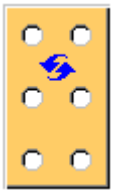
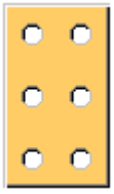
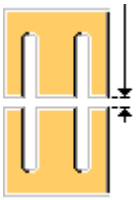
Bolt edge distances in shim plate

Define the bolt edge distances for shim plates. When these fields are empty, shim plates are of the same size as the end plate.



	Description	Default
1	Horizontal bolt edge distance in the shim plate.	30 mm
2	Vertical bolt edge distance in the shim plate.	30 mm

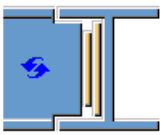
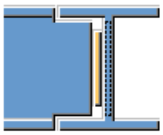
Shim plate shape

Option	Description
	Default Holes are based on the bolt group of the connection. AutoDefaults can change this option.
	Holes are based on the bolt group of the connection.
	Finger shim plate with horizontal slots. The plate can be installed from the right or the left side of the connection.
	Finger shim plate with vertical slots. The plate can be installed from the top of the connection.
	Two separate finger shim plates with horizontal slots.
	Two separate finger shim plates with vertical slots.

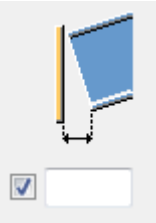
Tolerance

Define the tolerance for the slots in finger shim plates. The width of the slot is the bolt diameter + the tolerance. For two separate shim plates, also define the tolerance between the plates.

Shim plate position

Option	Description
	Default Shim plates are outside the main part. AutoDefaults can change this option.
	Shim plates are outside the main part.
	Shim plates are inside the main part.

Gap size



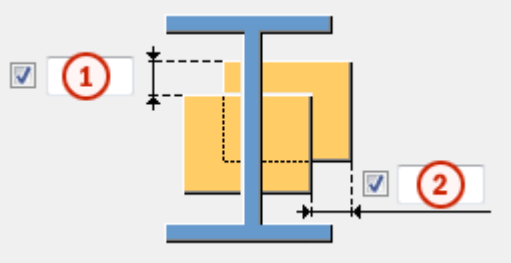
Define the limit value for the gap between the end plate and the secondary beam. Use this when the beam is slightly curved or sloped to decide if the end angle is so small that the beam end can be straight.

If the actual gap is smaller than this value, the end of the beam is left straight.

If the actual gap is larger than this value, the end of the beam is fitted to the end plate.

Shim plate and end plate positions

Define the position of the shim plates and the end plate in the second secondary beam. The plates move in relation to the plates in the first secondary beam. By default, the plates in the second secondary beam are positioned so that holes are placed symmetrically. You may need to move the plates, for example, when connecting skew or curved secondary beams.



	Description
1	Define how much the plates are moved in vertical direction.
2	Define how much the plates are moved in horizontal direction.

Haunch tab

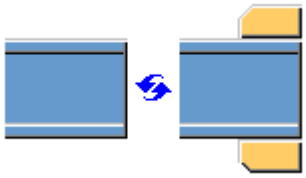
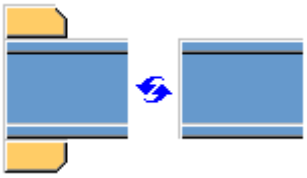
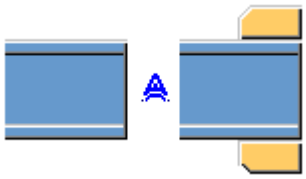
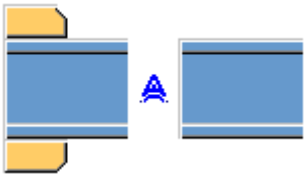
Use the **Haunch** tab to control the haunch plate creation and chamfers in the secondary beam flanges.

Haunch plates

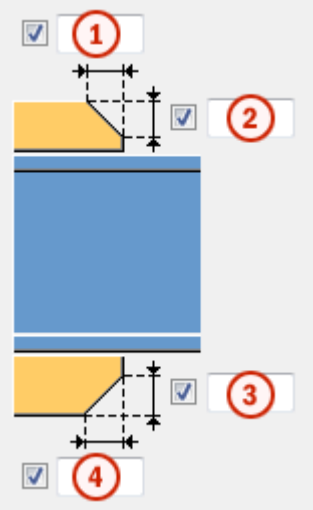
Option	Description
Top plate, **Top plate 2**	Top haunch plate thickness, width and height.
Bottom plate, **Bottom plate 2**	Bottom haunch plate thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Haunch plate creation

Option for the second secondary beam	Option for the first secondary beam	Description
		Default Top and bottom haunch plates are created, if needed. AutoDefaults can change this option.
		Automatic Top or bottom haunch plate or both are created, if needed.
		Top and bottom haunch plates are always created. To create a single plate, enter `0` in the thickness (**t**) field for the haunch plate you do not need (top or bottom plate).
		Haunch plates are not created.

Haunch plate chamfers



	Description
1	Width of the top haunch plate chamfer.
2	Height of the top haunch plate chamfer.
3	Height of the bottom haunch plate chamfer.
4	Width of the bottom haunch plate chamfer.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beams and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

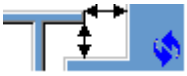
Automatic notching options affect both the top and the bottom flange.

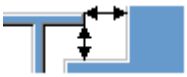
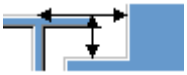
Notch shape

Automatic notching is switched on when you select a notch shape.

Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

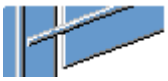
Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.

Option	Description
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

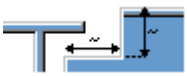
Enter the horizontal and vertical values for the cuts.

Flange cut shape

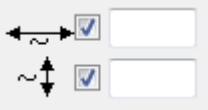
Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.



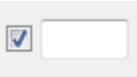
Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.



Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

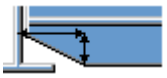
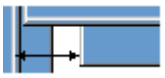
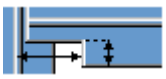
Side of flange notch

The side of flange notch defines on which side of the beam the notches are created.

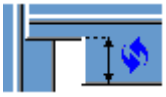
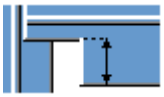
Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

Flange notch shape

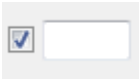
The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

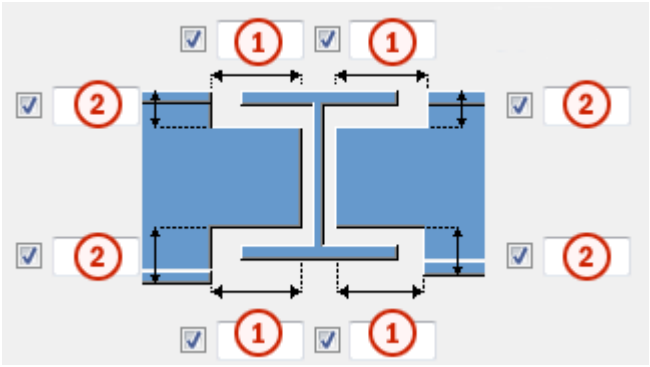
Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.



Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	`10 mm`
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest `5 mm`.

BCSA notch definition

Define whether the notch is created according to British Constructional Steelwork Association (BCSA) specifications.

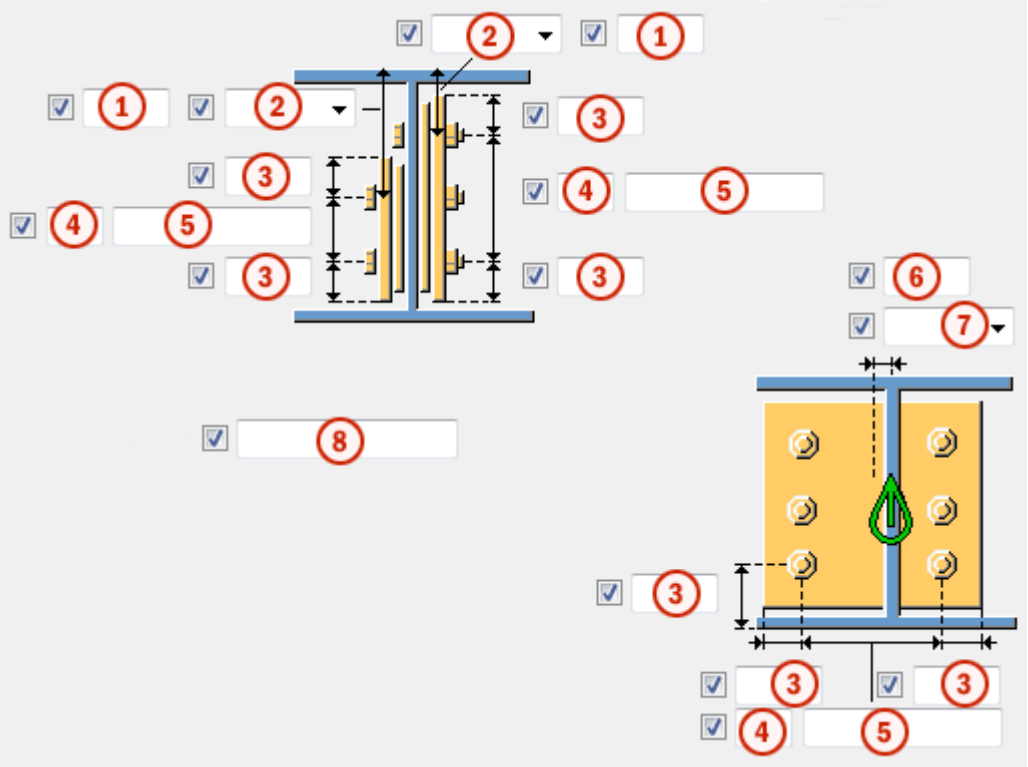
Option	Description
Default	Notch dimensions.
Yes	Creates a 50 mm notch for simple beam-to-beam connections.
No	Use the options on this **Notch** tab to define the notch dimensions.

Bolts tab

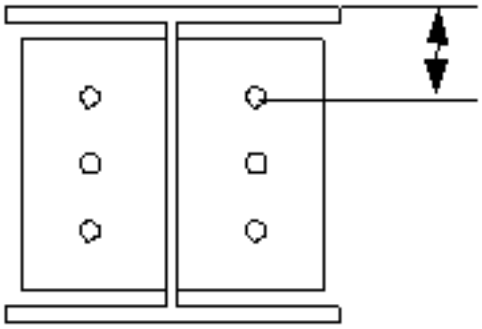
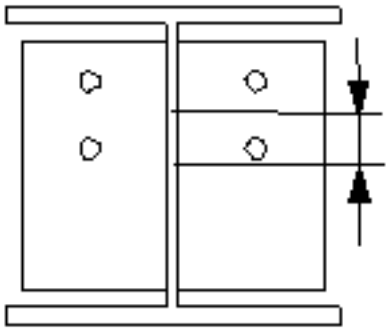
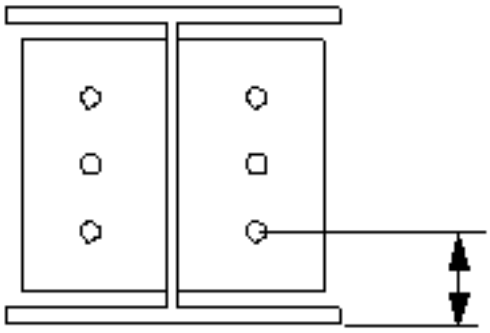
Use the **Bolts** tab to control the properties of bolts that connect the end plates to the main part.

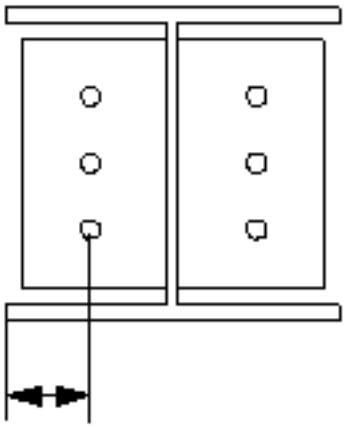
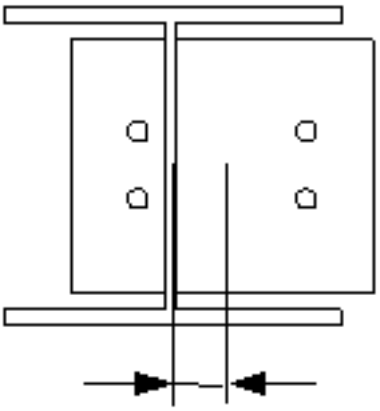
Bolt group dimensions

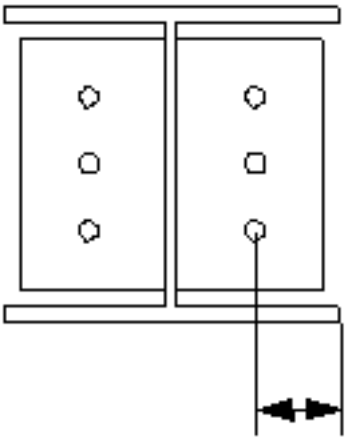
Bolt group dimensions affect the size of the end plates.



	Description
1	Dimension for vertical bolt group position.

	Description
2	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.

	Description
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Dimension for horizontal bolt group position.
7	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt.  • **Middle**: From the center line of the secondary part to the center line of the bolts. 

	Description
	• **Right**: From the right edge of the secondary part to the rightmost bolt. 
8	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

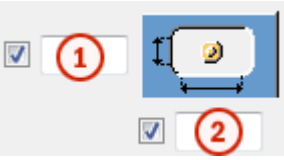
Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Bolt comment

You can define a bolt comment.

Slotted holes

You can define slotted, oversized, or tapped holes.

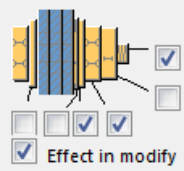


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

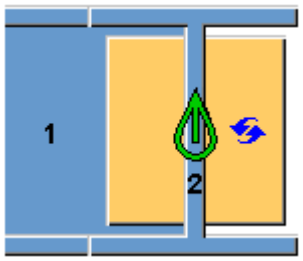
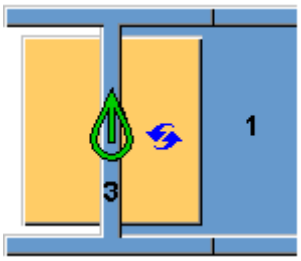
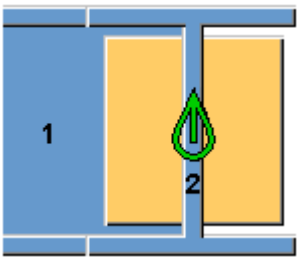
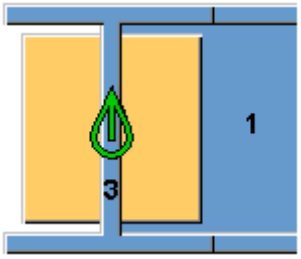
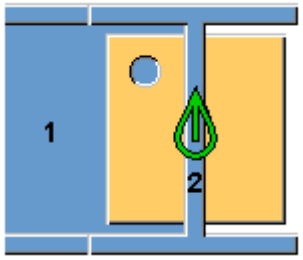
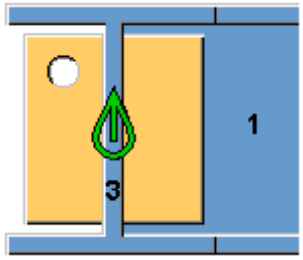
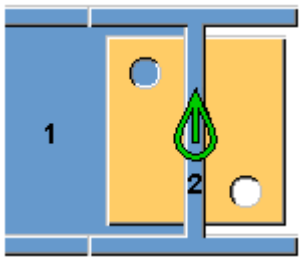
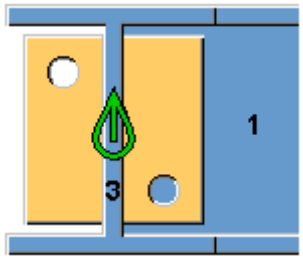
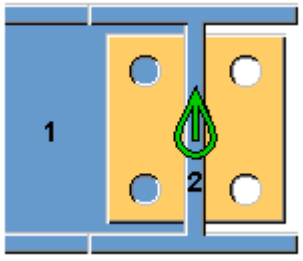
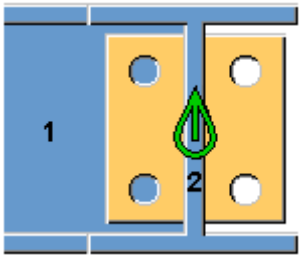
Holes tab

Use the **Holes** tab to control the galvanizing holes in the end plates.

Option	Description
Bolt standard	Select the bolt standard.
Bolt type	Select the bolt type to define the location where the bolts should be attached.
Read data from	You can select to use the `sinkholes.dat` definition file to specify the default values for horizontal and vertical offsets, and the diameters for upper and lower holes. The file is searched in the following order: Environment common system steel folder (`..\Environments\common\system\Steel`), model folder, `XS_FIRM`, `XS_PROJECT` and `XS_SYSTEM` folder. You can also select to define the holes in the component dialog box.

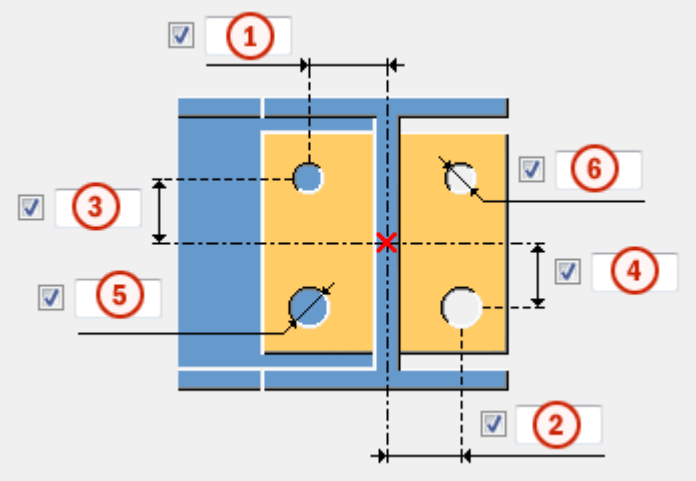
Number of holes

The center of a hole group is the middle point of the beam and the middle point of the haunch, if the haunch exists. The hole groups are composed of 0, 1, 2 or 4 holes.

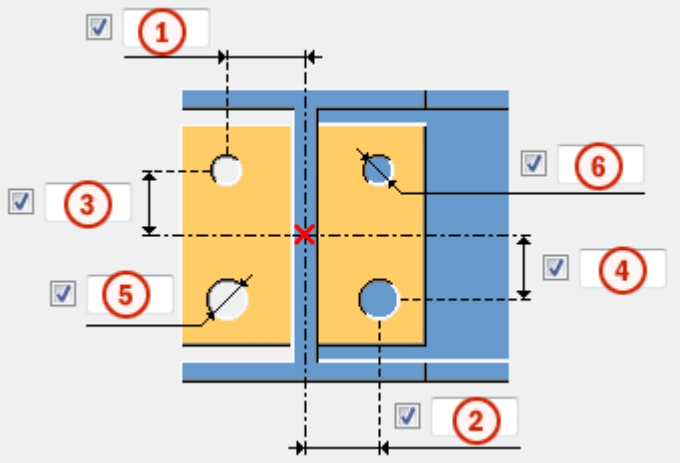
Option for the first secondary beam	Option for the second secondary beam	Description
		Default No holes AutoDefaults can change this option.
		No holes
		1 hole
		2 holes
		4 holes

Hole positions

Hole positions in the end plate of the first secondary beam.



Hole positions in the end plate of the second secondary beam.



	Description
1	Horizontal distance between the secondary beam center and the upper hole.
2	Horizontal distance between the secondary beam center and the lower hole.
3	Vertical distance between the secondary beam center and the upper hole.
4	Vertical distance between the secondary beam center and the lower hole.
5	Diameter of the lower hole.
6	Diameter of the upper hole.

General tab

Click the link below to find out more:

General tab

Design type tab

Click the link below to find out more:

Design type tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

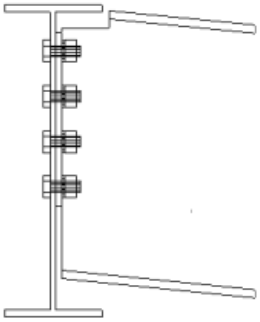
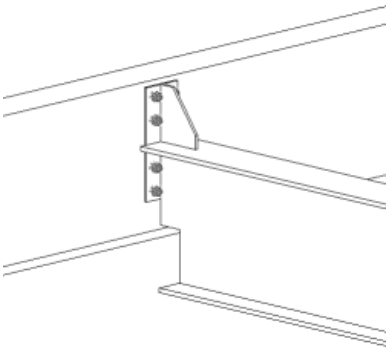
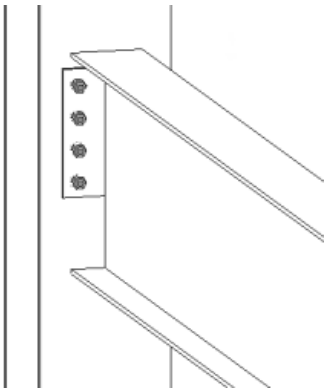
End plate (144)

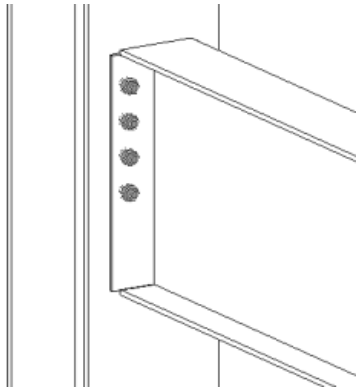
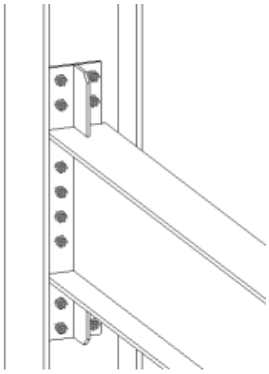
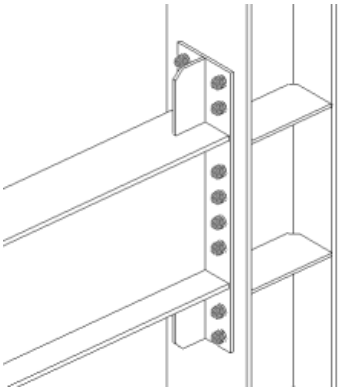
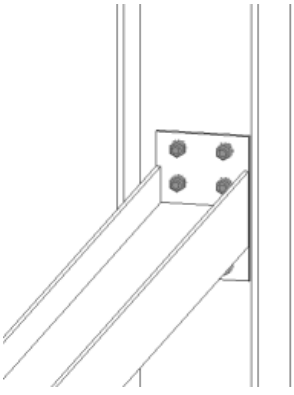
End plate (144) connects a beam to a column or two beams to each other using a bolted end plate. The secondary part can be leveled or sloped.

Objects created

- End plate
- Bent plate
- Shim plates (optional)
- Stiffeners (optional)
- Haunch plates (optional)
- Holes
- Bolts
- Seat angles
- Welds
- Cuts

Use for

Situation	Description
	Beam-to-beam end plate connection. The secondary part can either be leveled or sloped, or squared or skewed. There are several notching options.
	Beam-to-beam end plate connection. Extended plate with or without haunch plates.
	End plate connection to a column flange or web. The secondary part can either be leveled or sloped, or squared or skewed.

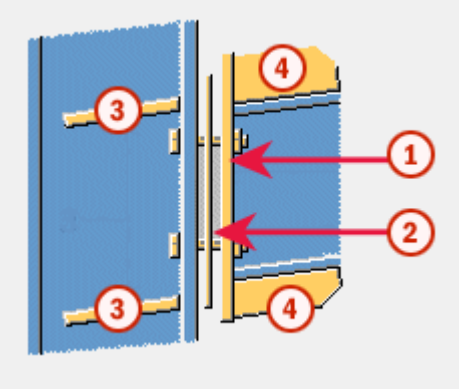
Situation	Description
	Full depth end plate connection to a column flange or web. The secondary part can either be leveled or sloped, or squared or skewed.
	End plate connection to a column web. Extended plate with haunch plates.
	End plate connection to a column flange with column stiffener.
	Column end plate connection. The secondary part is rotated.

Selection order

1. Select the main part (column or beam).
2. Select the secondary part (beam).

   The connection is created automatically.

Part identification key



	Part
1	End plate
2	Shim plate
3	Stiffener
4	Haunch plate

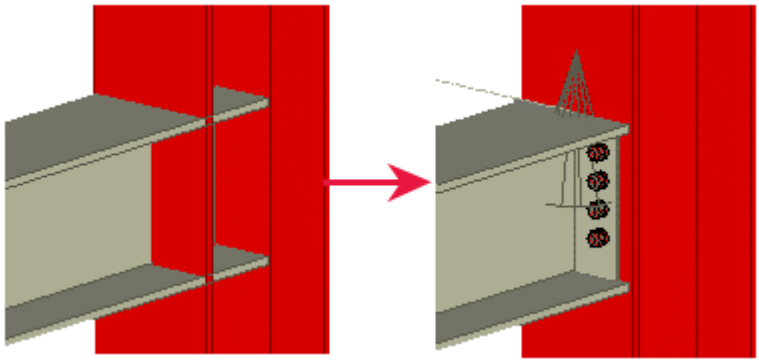
NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Example: Add an end plate using End plate (144)

In this example, you will connect a beam to a column using an end plate connection. **End plate (144)** connects two beams, or a beam to a column, using a bolted end plate.

1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. Type `144` in the search box.
3. Double-click **End plate (144)** to open the component properties.
4. Click **Apply** to add the component using the default properties.
5. Select the main part (column).
6. Select the secondary part (beam).

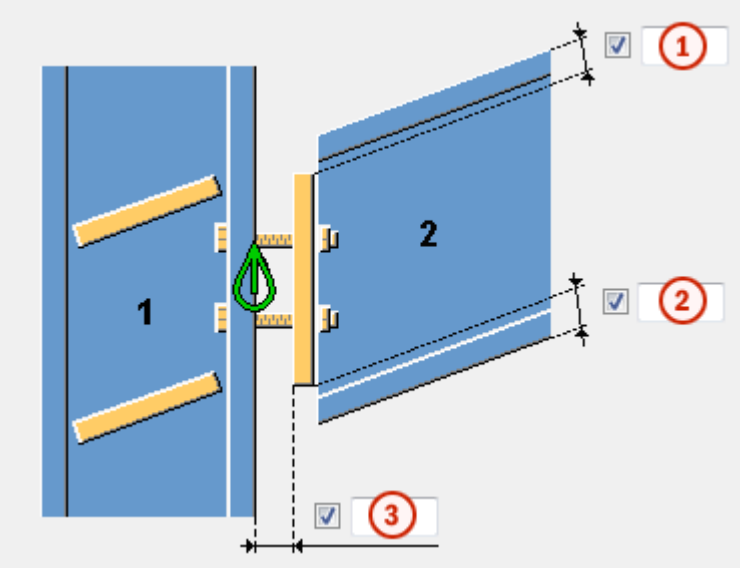
Tekla Structures automatically adds the connection when you select the beam.



Picture tab

Use the **Picture** tab to control the position of the end plate.

End plate position



	Description	Default
1	End plate upper edge distance from the top of the secondary beam. A positive value moves the top position closer to the beam center and thus decreases the plate size. Negative values increase the plate size.	10 mm
2	End plate lower edge distance from the bottom of the secondary beam.	

	Description	Default
3	Gap between the shim plate and the main part. If the shim plate is not used, the defined gap is created between the end plate and the main part.	0

Plates tab

Use the **Plates** tab to control the size of the end plate, bent plate, and shim plates.

Plates

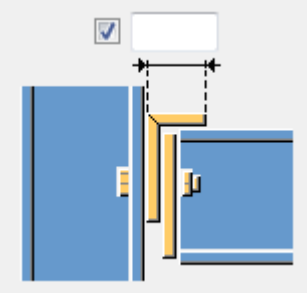
Option	Description	Default
End plate	End plate thickness, width and height. The width and height are defined by the bolt group edge distances.	thickness = 10 mm
Folded Plate	Bent plate thickness, width and height. The plate is created only if the plate thickness is given.	thickness = 0 width = bolt horizontal spacing value - (1.5*the bolt diameter) height = 200 mm
Fitting plate 1, 2, 3	Shim plate thickness. The plate is created only if the plate thickness is given. You can define up to three different shim plates.	0
Number of fitting plates	Number of shim plates for each thickness.	By default, 1 plate is created.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

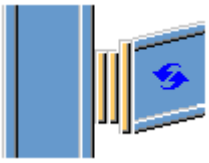
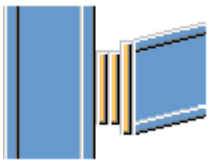
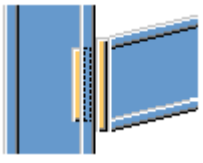
Option	Description	Default
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Bent shim plate length

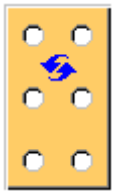
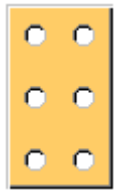
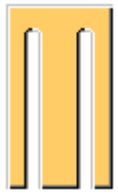
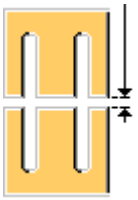
Define the horizontal length of the bent shim plate. The default is `10*bent plate thickness`.



Shim plate position

Option	Description
	Default Shim plates are created outside the main part. AutoDefaults can change this option.
	Shim plates are created outside the main part.
	Shim plates are created inside the main part.

Shim plate shape

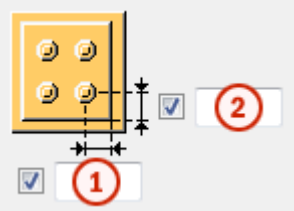
Option	Description
	Default Holes are based on the bolt group of the connection. AutoDefaults can change this option.
	Holes are based on the bolt group of the connection.
	Finger shim plate with horizontal slots. The plate can be installed from the right or the left side of the connection.
	Finger shim plate with vertical slots. The plate can be installed from the top of the connection.
	Two separate finger shim plates with horizontal slots.
	Two separate finger shim plates with vertical slots.

Tolerance

Define the tolerance for the slots in finger shim plates. The width of the slot is the bolt diameter + the tolerance. For two separate shim plates, also define the tolerance between the plates.

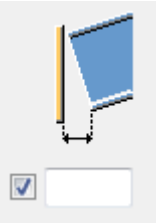
Bolt edge distances in shim plate

Define the bolt edge distances for shim plates. When these fields are empty, shim plates are of the same size as the end plate.



	Description	Default
1	Horizontal bolt edge distance in the shim plate.	30 mm
2	Vertical bolt edge distance in the shim plate.	30 mm

Gap size



Define the limit value for the gap between the end plate and the secondary beam. Use this when the beam is slightly curved or sloped to decide if the end angle is so small that the beam end can be straight.

If the actual gap is smaller than this value, the end of the beam is left straight.

If the actual gap is larger than this value, the end of the beam is fitted to the end plate.

Stiffeners tab

Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position, and type.

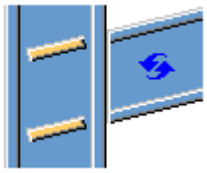
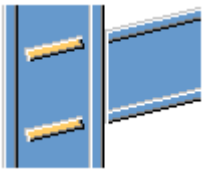
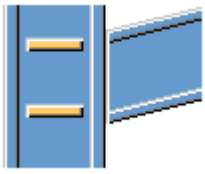
Stiffener plate dimensions

Option	Description
Top NS	Top near side stiffener thickness, width and height.
Top FS	Top far side stiffener thickness, width and height.
Bottom NS	Bottom near side stiffener thickness, width and height.

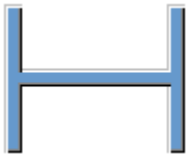
Option	Description
Bottom FS	Bottom far side stiffener thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Stiffener orientation

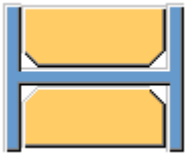
Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are parallel to the secondary part.
	Stiffeners are perpendicular to the main part.

Stiffener creation

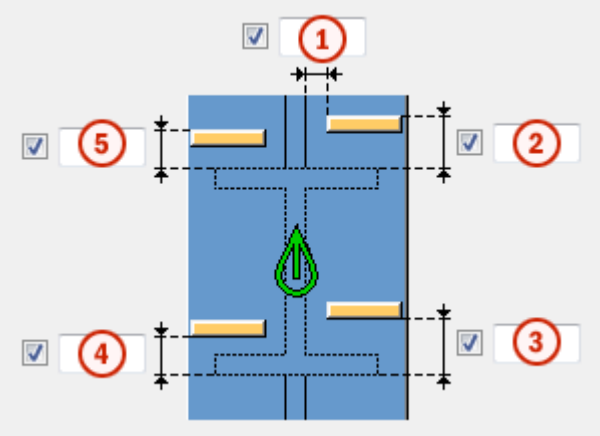
Option	Description
	Default Stiffeners are created. AutoDefaults can change this option.
	Automatic Stiffeners are created when necessary.
	No stiffeners are created.
	Stiffeners are created.

Stiffener shape

Option	Description
	Default Line chamfered stiffener plates AutoDefaults can change this option.
	Automatic Line chamfered stiffener plates
	Square stiffener plates Stiffener plates with a gap for the main part web rounding

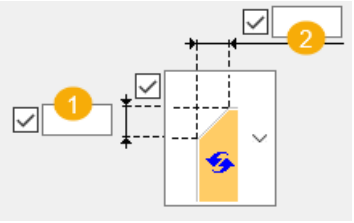
Option	Description
	Line chamfered stiffener plates

Stiffener positions



	Description
1	Size of the gap between the stiffener and the beam web edge.
2	Size of the gap between the top near side stiffener and the beam flange edge.
3	Size of the gap between the bottom near side stiffener and the beam flange edge.
4	Size of the gap between the bottom far side stiffener and the beam flange edge.
5	Size of the gap between the top far side stiffener and the beam flange edge.

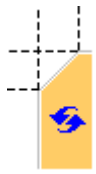
Chamfer dimensions



	Description	Default
1	Vertical dimension of the chamfer.	10 mm

	Description	Default
2	Horizontal dimension of the chamfer.	`10 mm`

Chamfer type

Option	Description
	Default. Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Haunch tab

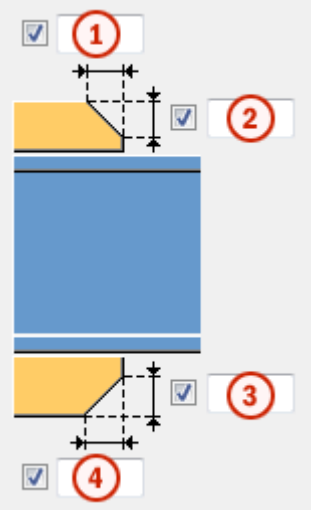
Use the **Haunch** tab to control the haunch plate creation and chamfers in the secondary beam flanges.

Haunch plates

Option	Description
Top plate	Top haunch plate thickness, width and height.
Bottom plate	Bottom haunch plate thickness, width and height.

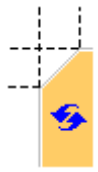
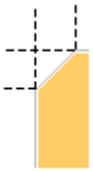
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Haunch plate chamfers

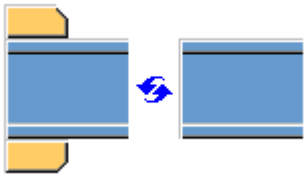
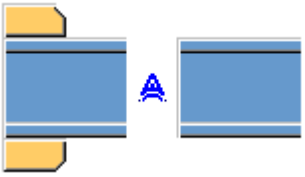


	Description
1	Width of the top haunch plate chamfer.
2	Height of the top haunch plate chamfer.
3	Height of the bottom haunch plate chamfer.
4	Width of the bottom haunch plate chamfer.

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Haunch plate creation

Option	Description
	Default Top and bottom haunch plates are created, if needed. AutoDefaults can change this option.
	Automatic Top or bottom haunch plate or both are created, if needed.

Option	Description
	Top and bottom haunch plates are created. To create a single plate, enter `0` in the thickness (**t**) field for the plate you do not need (top or bottom plate).
	Haunch plates are not created.

Notch tab

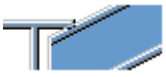
Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

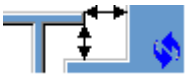
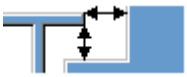
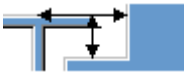
Automatic notching options affect both the top and the bottom flange.

Notch shape

Automatic notching is switched on when you select a notch shape.

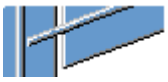
Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

Enter the horizontal and vertical values for the cuts.

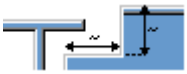
Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.

Option	Description
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.

Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.

Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

Side of flange notch

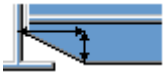
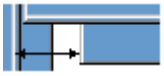
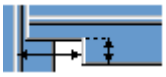
The side of flange notch defines on which side of the beam the notches are created.

Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

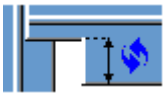
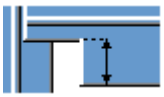
Flange notch shape

The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.

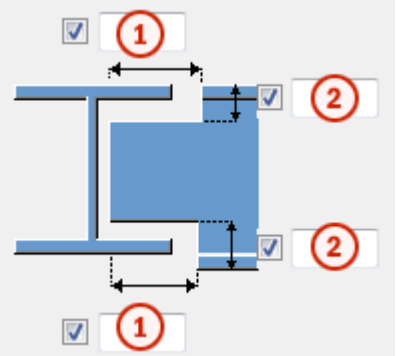
Option	Description
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.

Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	10 mm
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest 5 mm.

BCSA notch definition

Define whether the notch is created according to British Constructional Steelwork Association (BCSA) specifications.

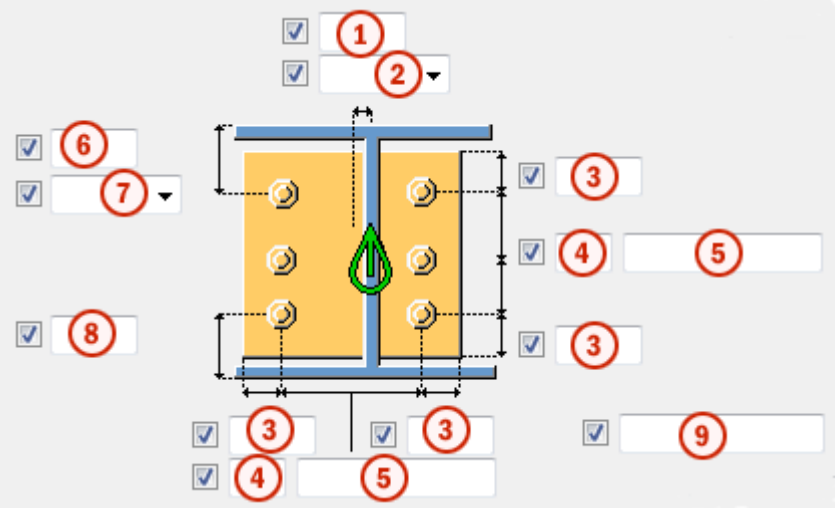
Option	Description
Default	Notch dimensions.
Yes	Creates a 50 mm notch for simple beam-to-beam connections.
No	Use the options on this **Notch** tab to define the notch dimensions.

Bolts tab

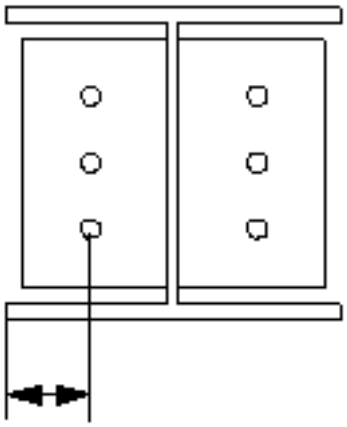
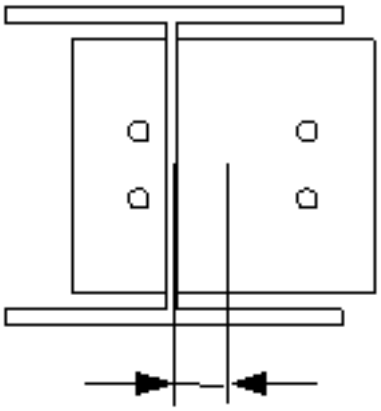
Use the **Bolts** tab to control the properties of the bolts that connect the end plate to the main part.

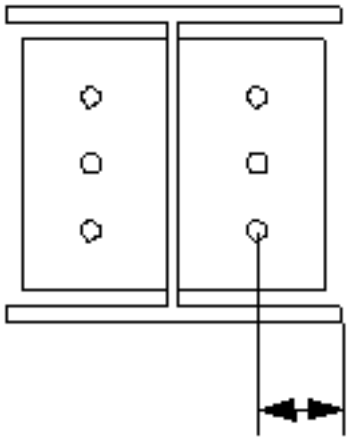
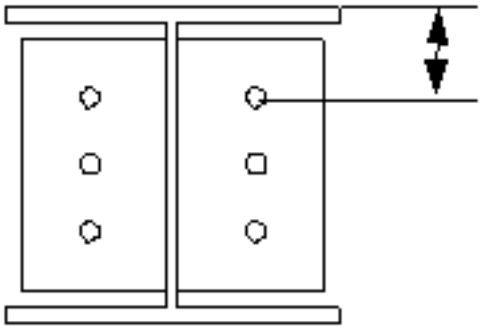
Bolt group dimensions

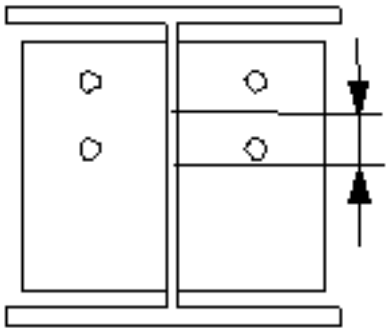
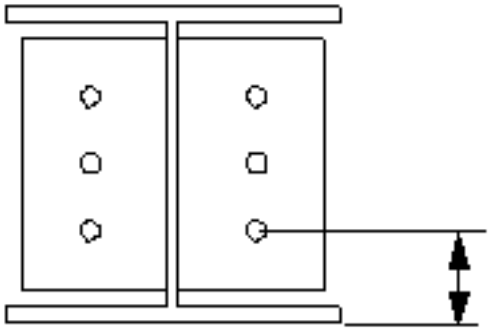
Bolt group dimensions affect the size of the end plate.



	Description
1	Dimension for horizontal bolt group position.

	Description
2	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt.  • **Middle**: From the center line of the secondary part to the center line of the bolts. 

	Description
	• **Right**: From the right edge of the secondary part to the rightmost bolt. 
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Dimension for vertical bolt group position.
7	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
8	Distance from the bottom of the beam to the lowest bolt.
9	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	

Option	Description	Default
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

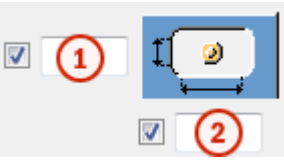
Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Bolt comment

You can define a bolt comment.

Slotted holes

You can define slotted, oversized, or tapped holes.



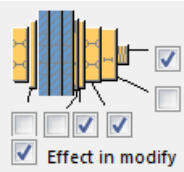
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options	

Option	Description	Default
	depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Bolting direction

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1
	Bolting direction 2

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.

Option	Description
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

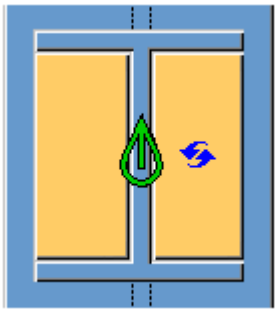
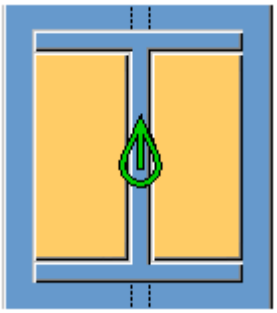
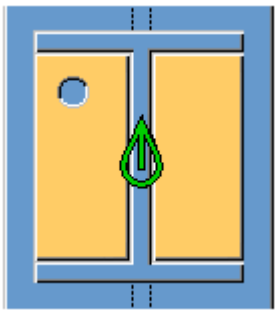
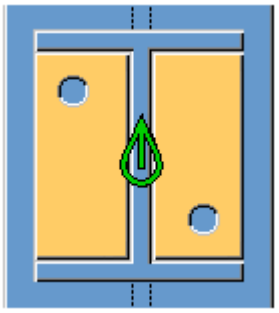
Holes tab

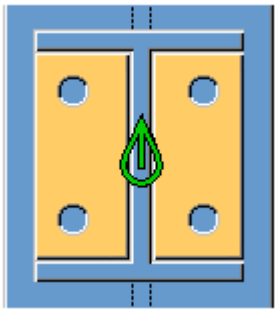
Use the **Holes** tab to control the galvanizing holes in the end plate.

Option	Description
Bolt standard	Select the bolt standard.
Bolt type	Select the bolt type to define the location where the bolts should be attached.
Read data from	You can select to use the `sinkholes.dat` definition file to specify the default values for horizontal and vertical offsets, and the diameters for upper and lower holes. The file is searched in the following order: Environment common system steel folder (`..\Environments\common\system\Steel`), model folder, `XS_FIRM`, `XS_PROJECT` and `XS_SYSTEM` folder. You can also select to define the holes in the component dialog box.

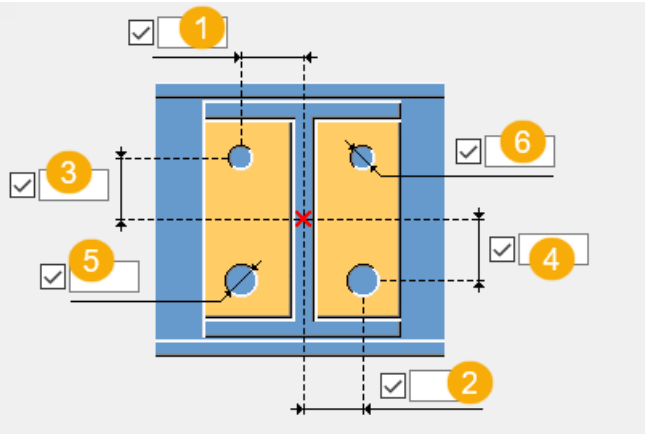
Number of holes

The center of a hole group is the middle point of the beam and the middle point of the haunch, if the haunch exists. The hole groups are composed of 0, 1, 2 or 4 holes.

Option	Description
	Default No holes AutoDefaults can change this option.
	No holes
	1 hole
	2 holes

Option	Description
	4 holes

Hole positions



	Description
1	Horizontal distance between the secondary beam center and the upper hole.
2	Horizontal distance between the secondary beam center and the lower hole.
3	Vertical distance between the secondary beam center and the upper hole.
4	Vertical distance between the secondary beam center and the lower hole.
5	Diameter of the lower hole.
6	Diameter of the upper hole.

Angle box tab

Use the **Angle box** tab to add a seat angle. The purpose of seat angles is to carry loads from the secondary beam.

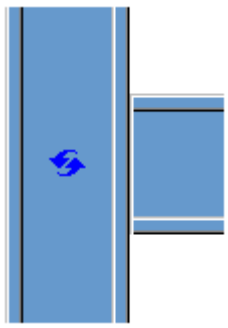
Seat angle

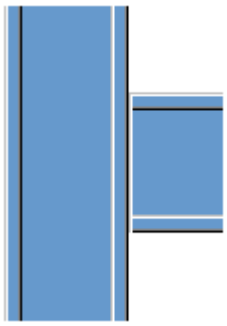
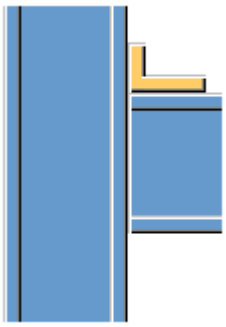
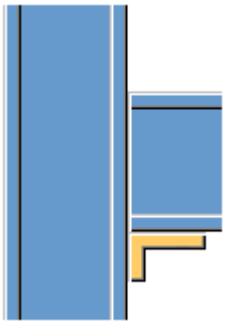
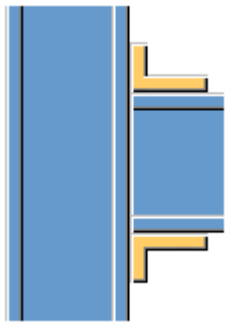
Option	Description
Top angle, **Bottom angle**	Seat angle thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

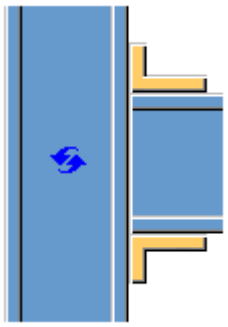
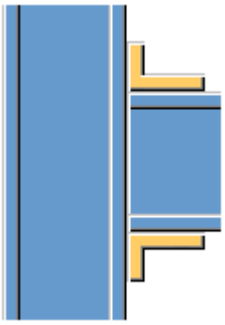
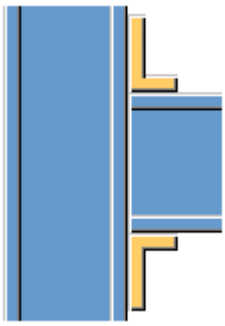
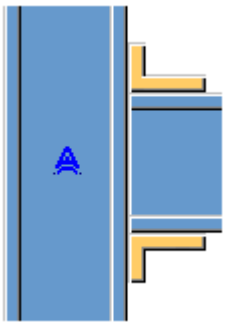
Seat angle position

Seat angles can be located on the top, bottom or both flanges of the secondary part.

Option	Description
	Default No seat angle is created. AutoDefaults can change this option.

Option	Description
	No seat angle is created.
	A seat angle is created at the top of the flange of the beam.
	A seat angle is created at the bottom of the flange of the beam.
	Seat angles are created on both sides of the flange of the beam.

Seat angle orientation

Option	Description
	Default The longer leg of the seat angle is connected to the secondary beam. AutoDefaults can change this option.
	The longer leg of the seat angle is connected to the secondary beam.
	The longer leg of the seat angle is connected to the main part.
	Automatic The longer leg of the seat angle is connected to the part where bolts reach furthest from the angle seat corner.

General tab

Click the link below to find out more:

General tab

Design type tab

Click the link below to find out more:

Design type tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

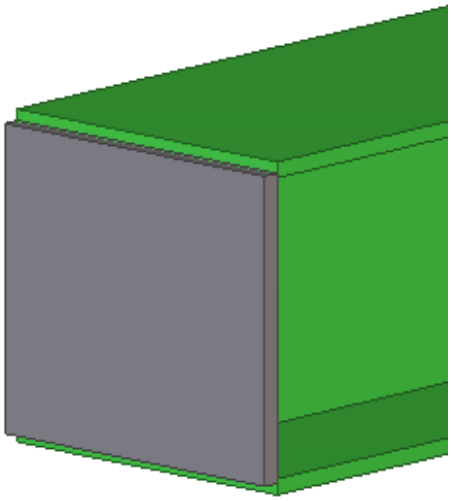
End plate detail (1002)

End plate detail (1002) creates an end plate that is welded to a beam end.

Objects created

- End plate

Use for

Situation	Description
	End plate at a beam end.

Selection order

1. Select the main part (beam).

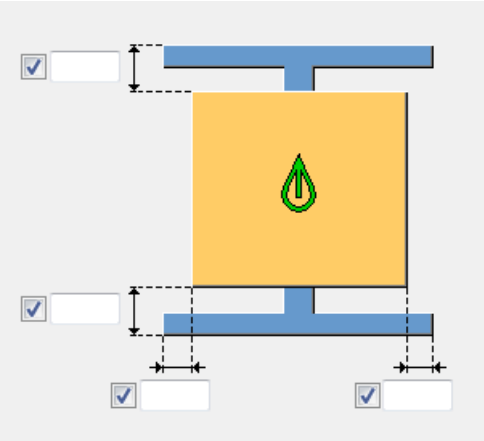
2. Pick a position.

   The detail is created automatically when the point is picked.

Picture tab

Use the **Picture** tab to control the end plate distances from the beam edges.

Positive values move the end plate closer to the beam axis and thus decrease the plate size. Negative values increase the plate size.



The default distances from the upper and the lower edge are `10 mm`.

The default distances from the right and the left edge are `0mm`.

For rectangular tube profiles all the distances are `3 mm` by default.

Parts tab

Use the **Parts** tab to control the end plate properties.

Option	Description
End plate	Define the end plate thickness, width and height. By default, the thickness is `1.5*beam web thickness` rounded up to the next plate thickness.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

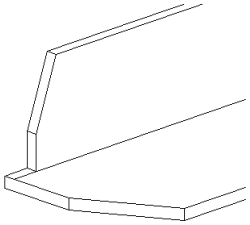
Angle cut (1057)

Angle cut (1057) creates fittings at the end of a profile that is of the angle type. The fittings are created at the end that is closer to the picked point. Two types of fittings are possible: one that chamfers the angle, or one that creates a polygon cut. The type is automatically decided according to the properties on the **Parameters** tab.

Objects created

- Fittings

Use for

Situation	Description
	Fittings at the end of a profile.

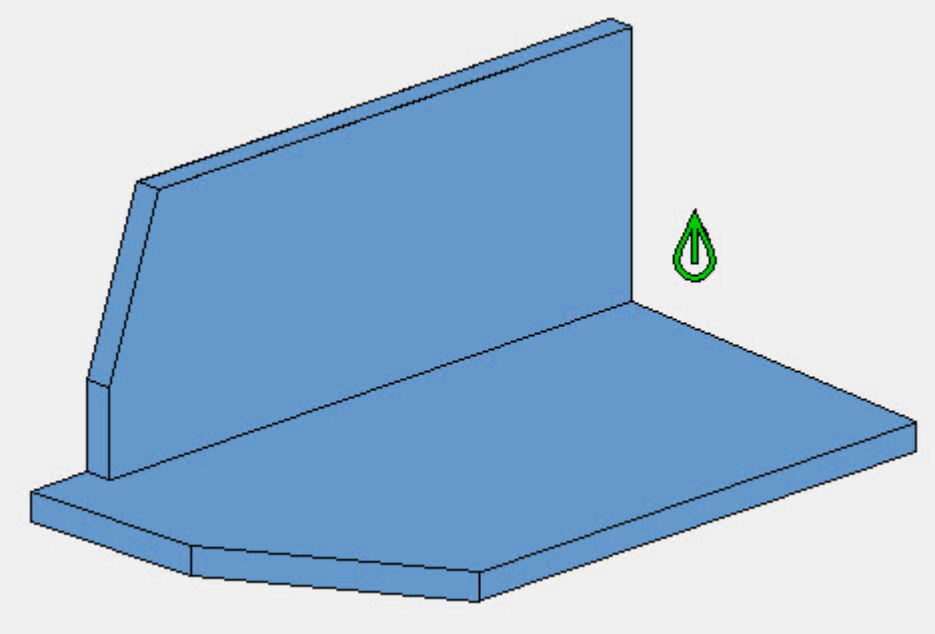
Selection order

1. Select the main part (angle profile).
2. Pick a position.

   The detail is created automatically when the point is picked.

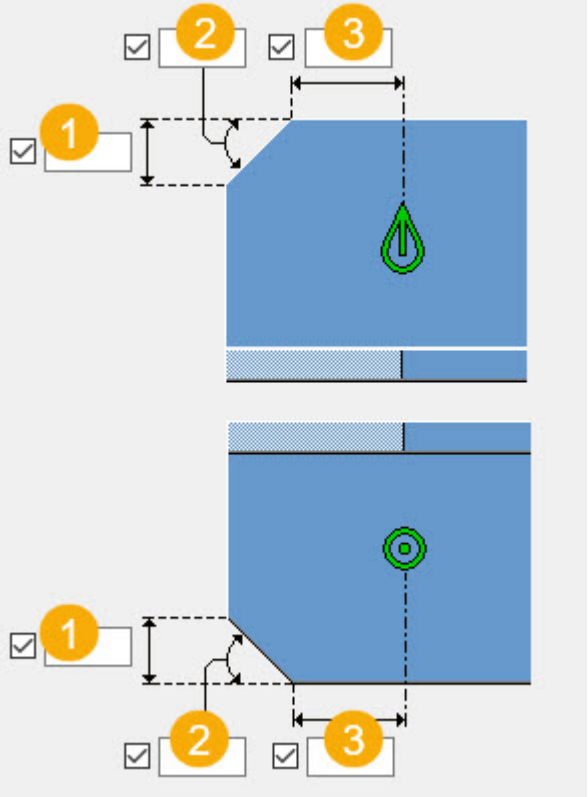
Picture tab

The **Picture** tab shows the angle cut.



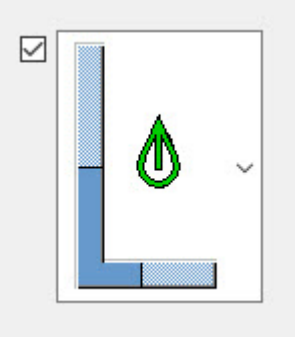
Parameters tab

Use the **Parameters** tab to define the fitting properties.



	Description	Default
1	Height of the flange cut. Defines the height of the cut. If the distance between the reference point and the profile end is so short that height of the cut cannot be achieved, then the profile is chamfered only.	The default value is half of the distance of the shorter edge.
2	Angle of the flange cut.	0 degrees
3	Offset distance for the flange. Defines the distance between the reference point and the start edge of the cut. It can have negative values as well.	0 mm

Select how the flange is cut. The options are to have a cut on both flanges, or to select which flange to cut.



General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

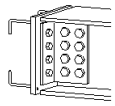
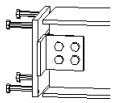
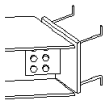
Cast-in plate (1069)

Cast-in plate (1069) creates a cast plate at the selected end of a beam. Studs, bolts, bent anchor bolts, or nail holes can be added to the plate.

Objects created

- Cast-in plate
- Shear tab
- Clip angle
- Bolts
- Studs
- Anchors
- Washer plates
- Welds

Use for

Situation	Description
	Bolted clip angle at the near and far side of the beam. The clip angle is anchored to the concrete using bent anchor bolts.
	Anchoring with four studs.
	Shear tab is bolted to a skewed beam.

Selection order

1. Select the main part (beam).
2. Pick a position near the end of the beam.

   Note that the point you pick determines from which end of the beam the plate will be positioned, it does not locate the plate itself.

   The connection is created automatically when you pick the position.

Part identification key

	Description
1	Cast-in plate
2	Clip angle or shear tab

Picture tab

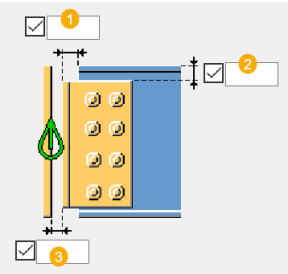
Use the **Picture** tab to define the connection type and dimensions.

Beam connection type

Option	Description
	Clip angle profile is used to connect the beam and the cast-in plate.

Option	Description
	Shear tab is used to connect the beam and the cast-in plate.

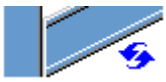
Dimensions



	Description	Default
1	Edge distance from the beam end to the shear tab or clip angle profile end.	1-2"
2	Edge distance from the beam top to the clip angle profile top. This option is only valid for clip angle profiles, not for shear tabs.	1-3/4"
3	Distance between the cast-in plate and the shear tab or clip angle profile.	0

Beam end cut

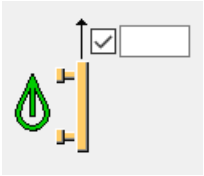
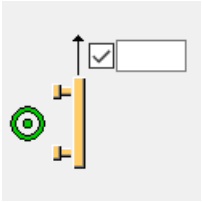
Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.
	Square Cuts the end of the secondary beam square.
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.

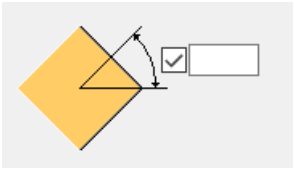
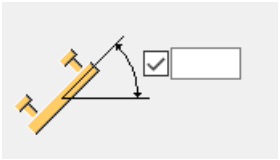
Cast-in plate alignment

Option	Description
	Default Cast-in plate is aligned and created on the beam plane. AutoDefaults can change this option.
	Cast-in plate is aligned and created on the beam plane.
	Cast-in plate is created in the global coordinate system.

Cast-in plate offset

Option	Description
	Cast-in plate height offset, measured in the plane of the plate.
	Cast-in plate out-of-plane offset, measured in the plane of the plate.

Cast-in plate rotation

Option	Description
	Cast-in plate rotation in degrees, measured in the plane of the plate.
	Cast-in plate out-of-plane rotation, in degrees, measured in the plane of the plate.

Parts tab

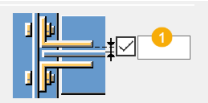
Use the **Parts** tab to define the part properties.

Parts

Option	Description
Cast-in plate	Thickness, width, and height of the cast-in plate.
Profile NS	Select the profile from the profile catalog.
Profile FS	Select the profile from the profile catalog.
Tab plate	Thickness, width, and height of the shear tab.
Plate washer	Thickness, width, and height of the washer plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Gap between clip angles



	Description
1	Gap between the secondary part web and clip angles. This only affects connections with two clip angles.

Clip angle position

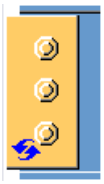
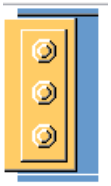
Option	Description
	Default One near side clip angle or shear tab is created. AutoDefaults can change this option.
	Automatic One near side clip angle or shear tab is created.
	Far side One far side clip angle or shear tab is created.
	Both sides Two clip angles or shear tabs are created on the near and far side.
	Near side One near side clip angle or shear tab is created.

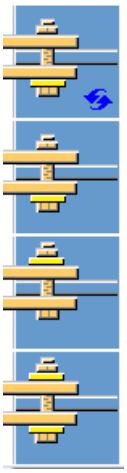
Clip angle leg direction

Far side	Near side	Description
		Default Not switched The long leg of the clip angle is connected to the secondary part. AutoDefaults can change this option.
		Not switched The long leg of the clip angle is connected to the secondary part.
		Switched The long leg of the clip angle is connected to the cast plate.

Plate washer

Define plate washers for bolts and select the plate washer side.

Option	Description
	Default No plate washer AutoDefaults can change this option.
	No plate washer
	One plate washer
	Individual square plate washers for each bolt
	Individual round plate washers for each bolt

Option	Description
	Select whether the plate washer is created for one shear tab or both shear tabs.

Studs/Anchors tab

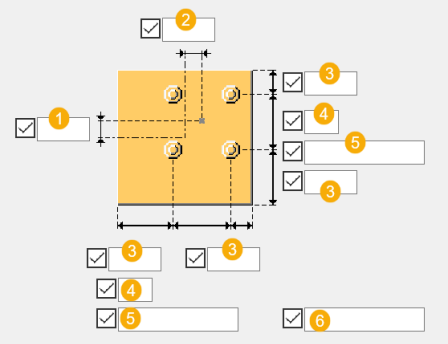
Use the **Studs/Anchors** tab to define whether studs or anchors are created, and their properties.

Properties

Option	Description
Stud size	Diameter of the stud.
Stud standard	Name of the stud.
Stud length	Length of the stud.
Site / **Workshop**	Location where the bolts should be attached.
Std/Anc	Select whether to create studs, anchors, or none of these.
Stud/Anc profile	Select the profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Bolt group dimensions

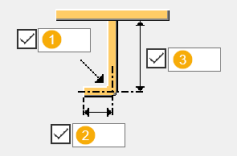


	Description
1	Vertical bolt group dimension from the center line of the plate.
2	Horizontal bolt group dimension from the center line of the plate.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Anchor bend direction

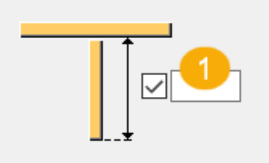
Option	Description
	Default Anchors are bent in. AutoDefaults can change this option.
	Anchors are bent in.
	Anchors are bent out.
	Anchors are not bent. Studs are created.

Bent anchor bolt length



	Description	Default
1	Radius of the bend.	3/8"
2	Bent anchor bolt length.	4"
3	Hook length.	1-1/2"

Stud length



	Description	Default
1	Define the stud length.	4"

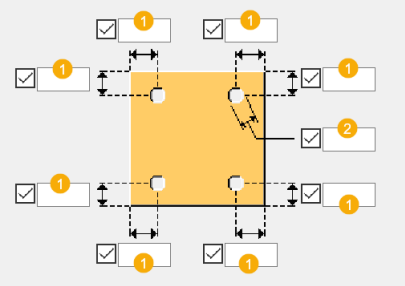
Nail holes tab

Use the **Nail holes** tab to define the nail hole locations and dimensions.

Nail hole locations

Option	Description
	Default No nail holes. AutoDefaults can change this option.
	No nail holes.
	Creates nail holes in the upper right and lower left corners of the plate.
	Creates nail holes in the upper left and lower right corners of the plate.
	Creates nail holes in all four corners of the plate.

Nail hole dimensions

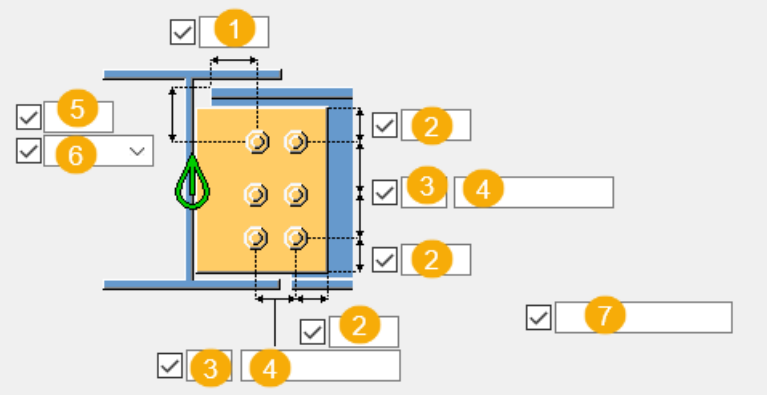


	Description
1	Nail hole edge distance.
2	Nail hole diameter.

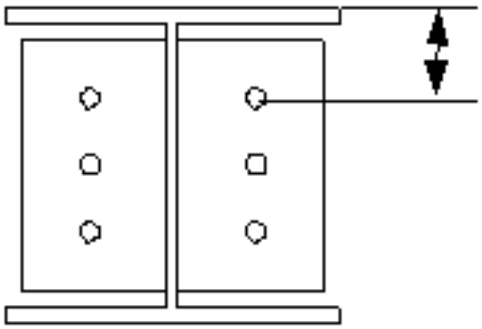
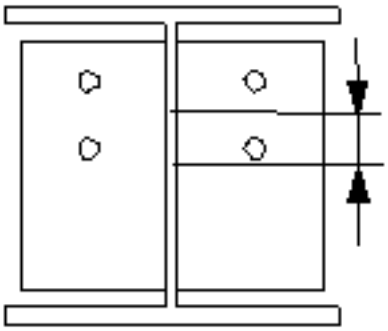
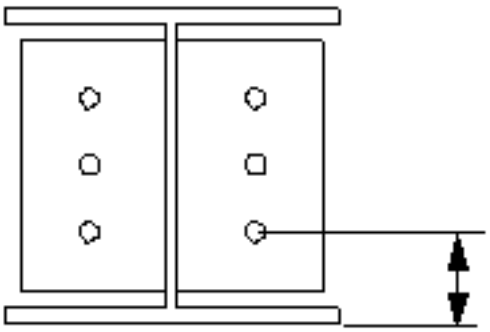
Bolts tab

Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position.

	Description
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

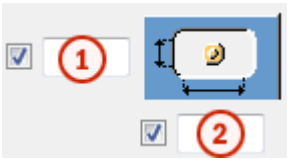
	Description
7	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



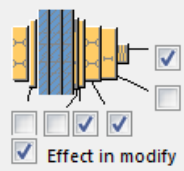
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered

Option	Description
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt group orientation

Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Staggered Bolts are staggered in the direction of the secondary part.
	Square Square bolt group is positioned horizontally.
	Sloped Square bolt group is sloped in the direction of the secondary part.

General tab

Click the link below to find out more:

General tab

Design type tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

5.5 Splice connections

This section introduces Tekla Structures splice connection components.

- Column splice (42) (page 1652)
- Diagonal splice (53) (page 1663)
- Splice connection (77) (page 1670)
- Column splice (132) (page 1690)

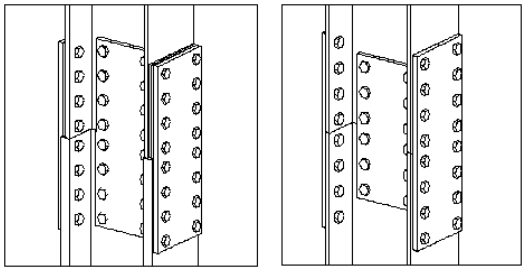
Column splice (42)

Column splice (42) connects two columns (or beams) using splice plates bolted to the flanges and webs of both parts. Shim plates are also created when the dimensions of the columns differ.

Objects created

- Web plates
- Flange plates
- Shim plates (optional)
- Bolts

Use for

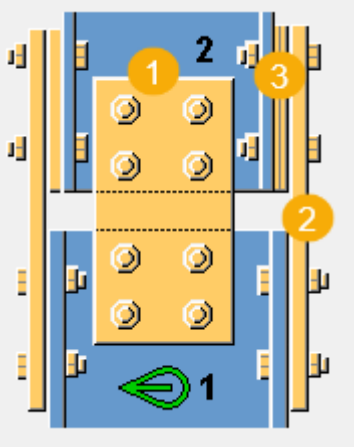
Situation	Description
	Splice plates bolted to the flanges and webs of both parts.

Selection order

1. Select the main part (beam or column).
2. Select the secondary part (beam or column).

   The connection is created automatically when the secondary part is selected.

Part identification key

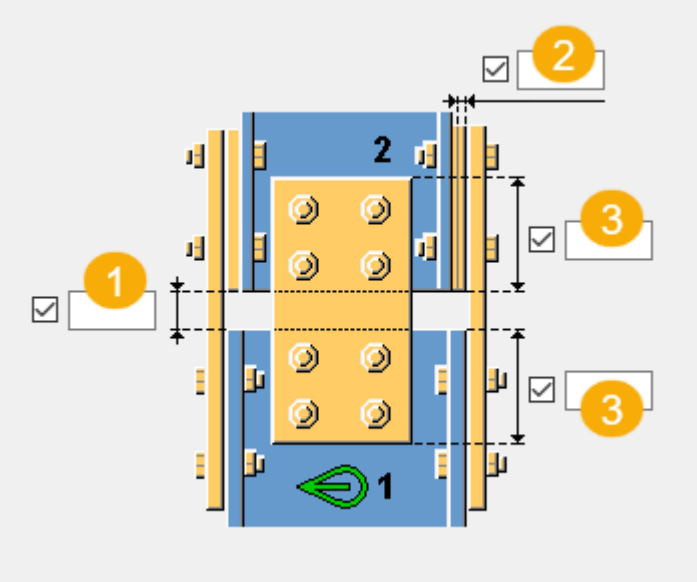


	Description
1	Web plate
2	Flange plate
3	Shim plate

Picture tab

Use the **Picture** tab to define the plate positions.

Dimensions



	Description	Default
1	Gap between the main part and the secondary part.	0 mm
2	Shim plate thickness	10 mm
3	Web plate dimensions on the main part and the secondary part.	

Part tab

Use the **Parts** tab to define the properties of the plates.

Plates

Option	Description
Web plates	Thickness, width and height of the web plates.
Flange plates	Thickness, width and height of the flange plates.

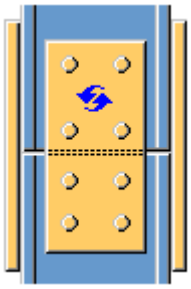
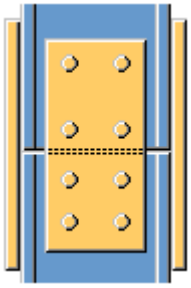
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

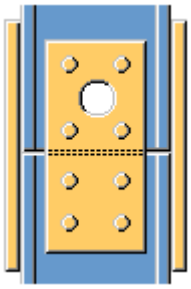
Option	Description	Default
	assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Parameters tab

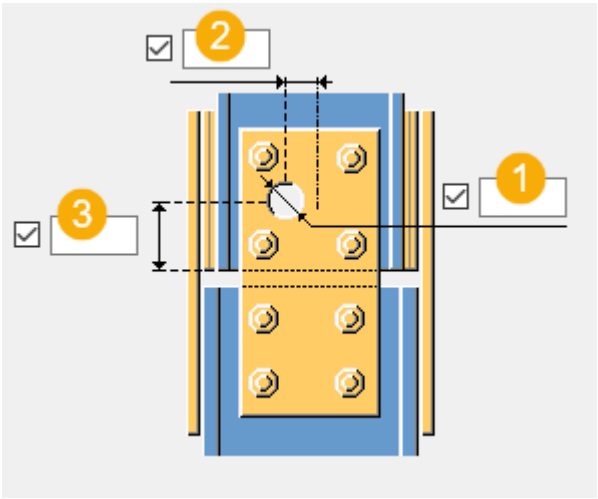
Use the **Parameters** tab to define the lifting hole creation, dimensions, and web plate creation.

Lifting hole creation

Option	Description
	Default Lifting hole is not created. AutoDefaults can change this option.
	Lifting hole is not created.

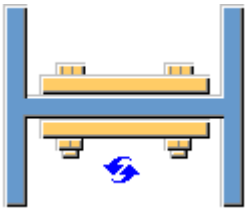
Option	Description
	Lifting hole is created.

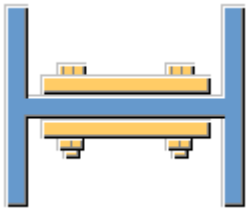
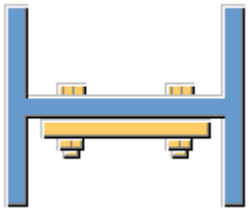
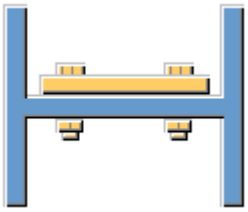
Lifting hole dimensions



	Description
1	Diameter of the lifting hole.
2	Horizontal dimension for the lifting hole offset from the web plate center line.
3	Vertical dimension from the lifting hole center to the part edge.

Web plates

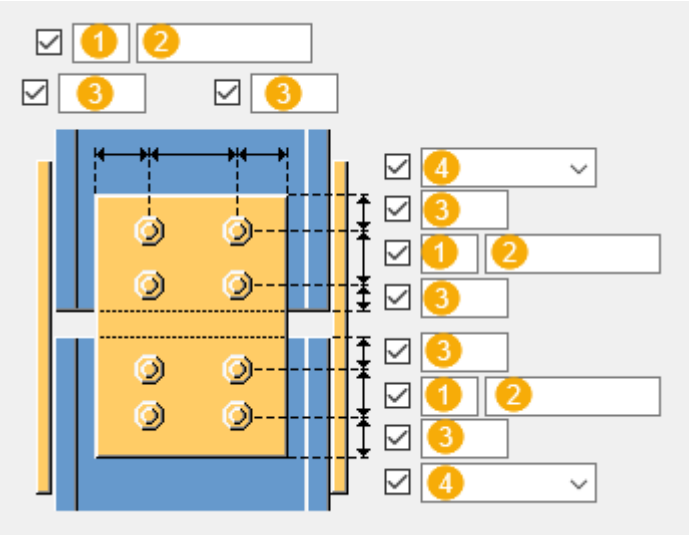
Option	Description
	Default Both web plates are created. AutoDefaults can change this option.

Option	Description
	Both web plates are created.
	Right side web plate is created.
	Left side web plate is created.

Web bolts tab

Use the **Web bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions



	Description
1	Number of bolts.

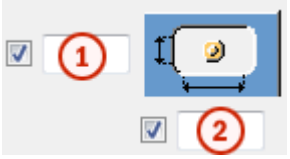
	Description
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Select the bolt type.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.



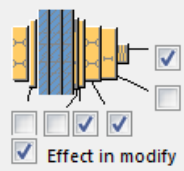
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

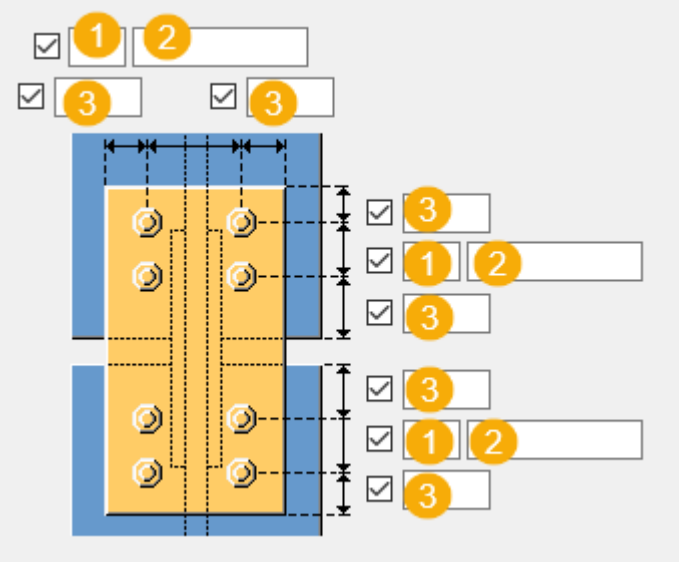
Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered

Option	Description
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Flange bolts tab

Use the **Flange bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions



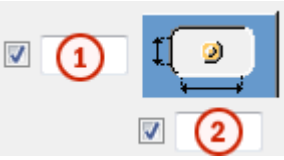
	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.



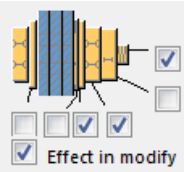
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options	

Option	Description	Default
	depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

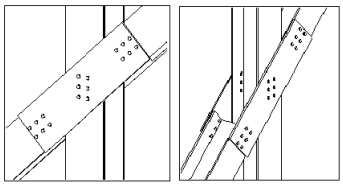
Diagonal splice (53)

Diagonal splice (53) connects two beams to a column using connection plates bolted to the column and beam flanges.

Objects created

- Connection plates
- Bolts

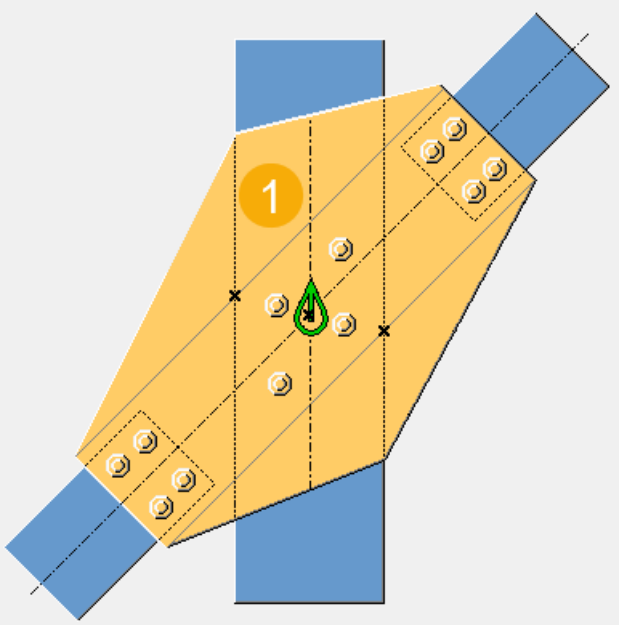
Use for

Situation	Description
	Beams are connected to a column with bolted connection plates.

Selection order

1. Select the main part (column).
2. Select the first secondary part (beam).
3. Select the second secondary part (beam).
4. Click the middle mouse button to create the connection.

Part identification key

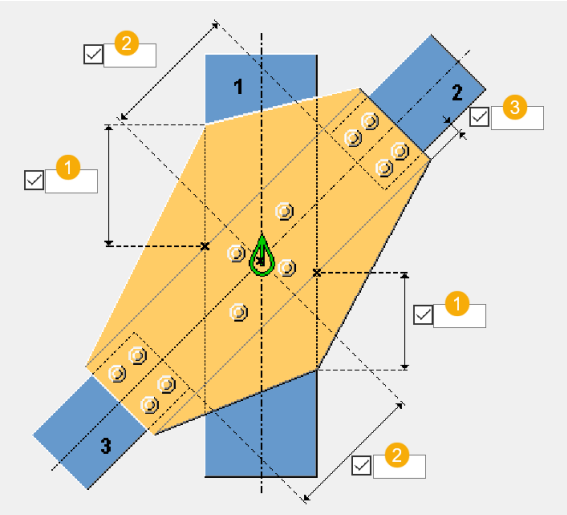


	Description
1	Connection plate

Picture tab

Use the **Picture** tab to define the geometry of the connection.

Dimensions



	Description	Default
1	Define a vertex to the connection plate at the lowermost and uppermost intersection point of the column edge and the plate upper or lower edge.	`0` Rectangular plate is created.
2	Define the distance to the first selected and second selected secondary part from the column center line to the secondary part end.	If the beams are perpendicular to the column, the default value for this distance is equal to the column flange width. Otherwise, the default distance is calculated so that the beam does not collide with the column.
3	Define the plate tolerance. You can use this dimension to increase the plate width.	`10 mm`

Parts tab

Use the **Parts** tab to define the part properties.

Parts

Option	Description
Create as	Select whether to create a connection plate or a profile.
Plate	Thickness of the connection plate.
Profile	Select the profile from the profile catalog.
Spacer plate	Thickness, width, and height of the spacer plate. If the main part profile is higher than the profile of the diagonal parts, spacer plates can be created between the connection plate and the diagonal parts to fill the empty space. To do

Option	Description
	this, the diagonal parts should be centrally connected to the main part. By default, the thickness of the plate is equal to the width of the gap between the connection plate and the secondary beam. If the thickness is defined, the component creates as many plates as can be fitted in the gap.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

Use the **Parameters** tab to define the connection plate positions.

Twin profiles

If you are using twin profiles as secondary parts, select **Yes** in this option.

When selecting the secondary parts to this connection, select the outer part of the twin profiles.

Plate position

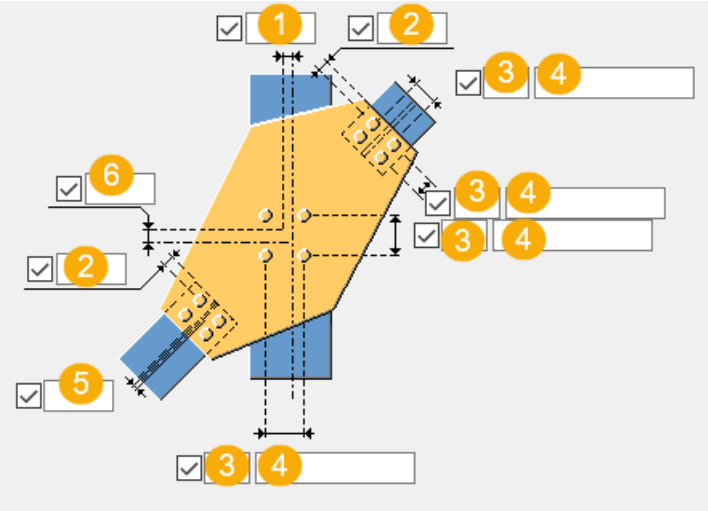
Option	Description
	Default Plates are created on both sides. AutoDefaults can change this option.

Option	Description
	Plate is created on the left side.
	Plates are created on both sides.
	Plate is created on the right side.
	Plates are created on both sides. Long bolts go through the column and beam flanges.

Bolts tab

Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Bolt group offset from the connection plate center line.

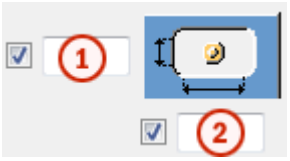
	Description
6	Dimension for vertical bolt group position.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



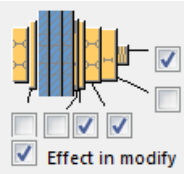
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	

Option	Description	Default
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2

Option	Description
	Staggered type 3
	Staggered type 4

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

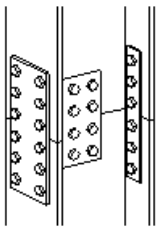
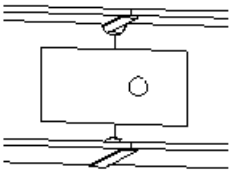
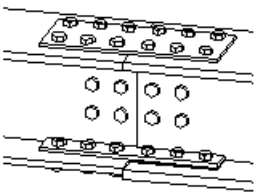
Splice connection (77)

Splice connection (77) creates a splice between two beams or columns. The splice can be bolted or welded, or bolted and welded. This connection also works with rotated and skewed beams, but only with I-shaped cross-sections.

Objects created

- Flange plates
- Web plates
- Deck plates
- Bolts
- Welds

Use for

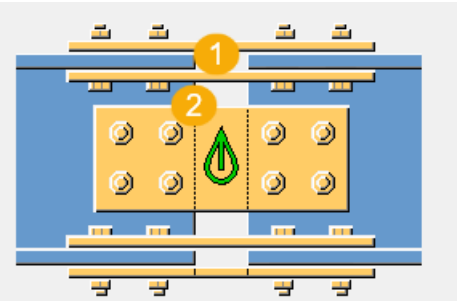
Situation	Description
	Plates are bolted.
	Plates are welded.
	Plates are bolted and welded.

Selection order

1. Select the main part.
2. Select the secondary part.

   The connection is created automatically when the secondary part is selected.

Part identification key

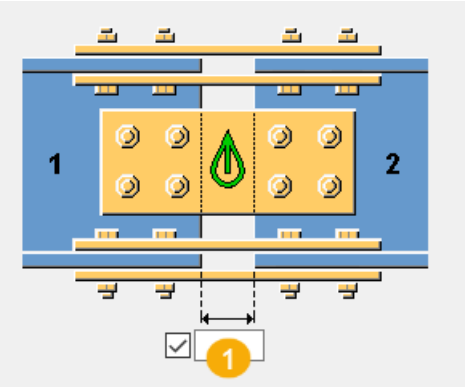


	Part
1	Flange plates
2	Web plates

Picture tab

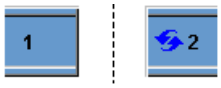
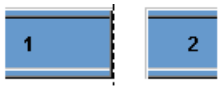
Use the **Picture** tab to define the gap between the main and secondary beam, and the gap type.

Gap dimension



	Description	Default
1	Gap between the main part and secondary part.	5 mm or ¼"

Gap type

Option	Description
	Default Center fit By default, both parts are cut equally (1/2 * entered gap). AutoDefaults can change this option.
	Center fit The main part and secondary parts are cut equally (1/2 * entered gap), and placed centrally.
	Offset fit Only the secondary part is cut.
	Offset fit Only the main part is cut.

Parts tab

Use the **Parts** tab to define the properties of flange plates and spacers.

Flange plates

Option	Description
Top ext flange pl **Top int flange pl** **Btm ext flange pl** **Top int flange pl**	Flange plate thickness, width, and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Create web connection as

Option	Description
Plates	Plate thickness, width, and height.
Profiles	Select the profile from the profile catalog.

Spacers

The flange and web spacers are used when the main and secondary beam flange or web thicknesses differ.

Option	Description
Flange spacers **Top** **Bottom**	Flange spacer thickness.
Create external flange spacers	Select whether external flange spacers are created.
Web spacers	Web spacer thickness.

Parameters tab

Use the **Parameters** tab to define the location and diameter of the lifting hole, the type of bolts, and loose part and splice plate assembly.

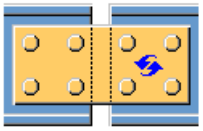
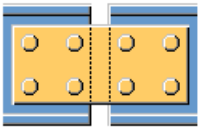
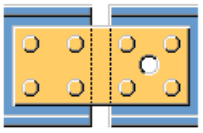
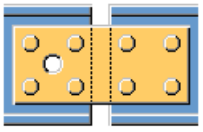
Bolts and assemblies

Option	Description
Workshop bolts	• **None** - All bolts are attached at the building site. • **Primary member** - The bolts in the main part are attached at the workshop, and the bolts in the secondary part are attached at the building site. • **Secondary member** - The bolts in the secondary part are attached at the workshop, and the bolts in the main part are site bolts.
Loose parts assembly	• **Primary member** - Creates a loose assembly between the spacers and the main part. • **Secondary member** - Creates a loose assembly between the spacers and the secondary part. • **Both** - Creates a loose assembly between the spacers and the main part as well as between the spacers and the secondary part.
Splice plate assembly	• **None** - No assembly is created either between the splice plates or the main and secondary beams. This is the default value. • **Primary member** - Creates an assembly between the splice plates and the main beam.

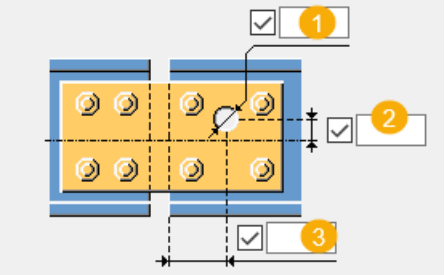
Option	Description
	• **Secondary member** - Creates an assembly between the splice plates and the secondary beam. • **Both** - Creates an assembly between the splice plates and both the main and secondary beams. Note that the assembly is created by welding the splice plates to the appropriate beams.

Lifting hole creation

Select to which part the lifting hole is created: plates, beam, or to both the plates and the beam.

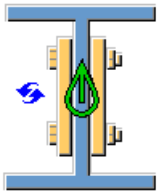
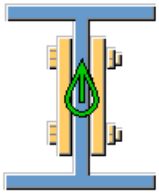
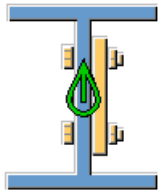
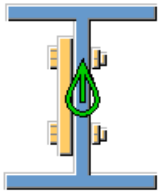
Option	Description
	Default Lifting hole is not created. AutoDefaults can change this option.
	Lifting hole is not created.
	Lifting hole is created in the secondary part side.
	Lifting hole is created in the main part side.

Lifting hole dimensions

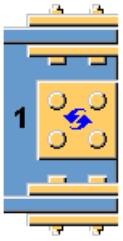
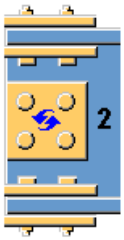


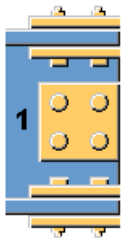
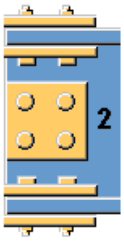
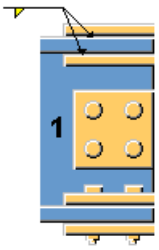
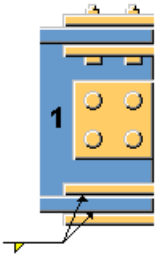
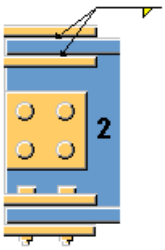
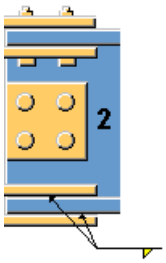
	Option
1	Define the diameter of the lifting hole.
2	Define the vertical location of the lifting hole from the center line of the web.
3	Define the horizontal location of the lifting hole from the end of the secondary beam.

Web plate creation

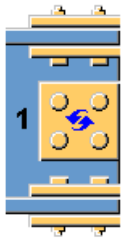
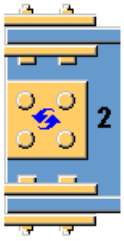
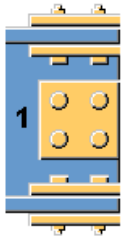
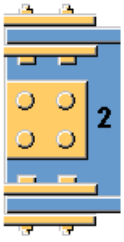
Option	Description
	Default Both web plates are created. AutoDefaults can change this option.
	Both web plates are created.
	Web plate is created on the front side.
	Web plate is created on the back side.

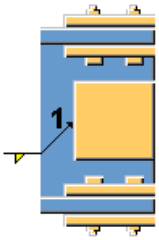
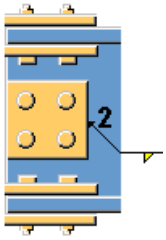
Attach flange plates to main and secondary parts

Option	Option	Description
		Default Flange plates are bolted to the main and secondary parts. AutoDefaults can change this option.

Option	Option	Description
		Flange plates are bolted to the main and secondary parts.
 	 	Flange plates are welded to the main and secondary parts.

Attach web plates to main and secondary parts

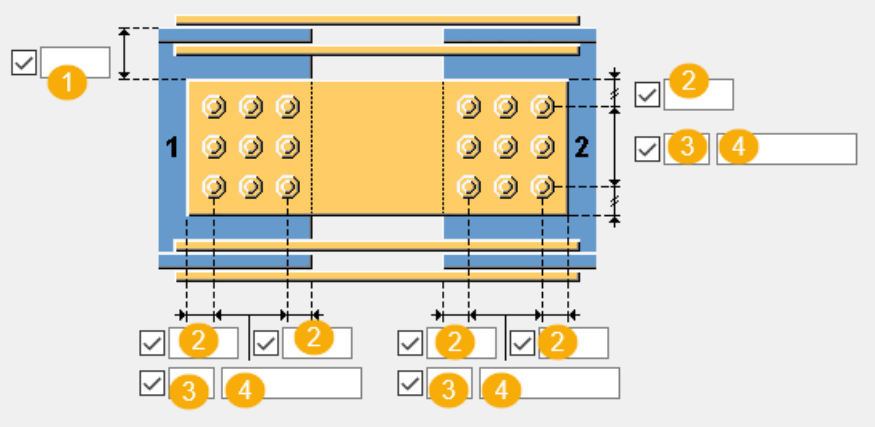
Option	Option	Description
		Default Web plates are bolted to the main and secondary parts. AutoDefaults can change this option.
		Web plates are bolted to the main and secondary parts.

Option	Option	Description
		Web plates are welded/ bolted to the main and secondary parts.

Web bolts tab

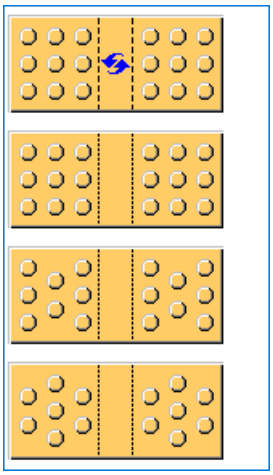
Use the **Web bolts** tab to define the web bolt properties.

Bolt group dimensions



	Description
1	Dimension for vertical bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Staggering of bolts

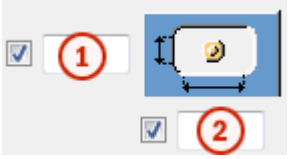
Option	Description
	Select the appropriate staggering option.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.

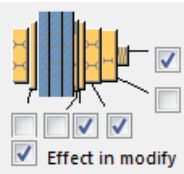


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

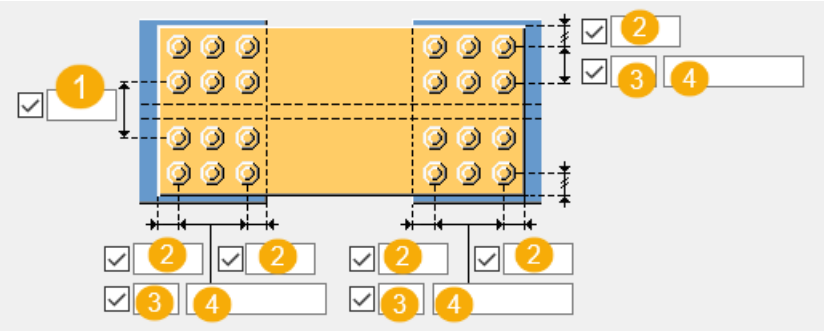
Bolting direction

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1
	Bolting direction 2

Top flange bolts / Bottom flange bolts tab

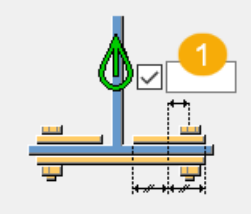
Use the **Top flange bolts** and **Botton flange bolts** tabs to define the flange bolt properties. On the **Bottom flange bolts** tab, you can select to use the top flange bolt default settings for the bottom bolts.

Bolt group dimensions



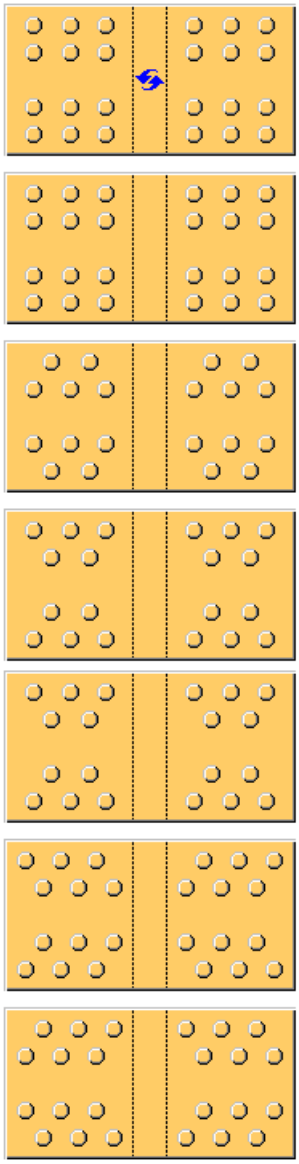
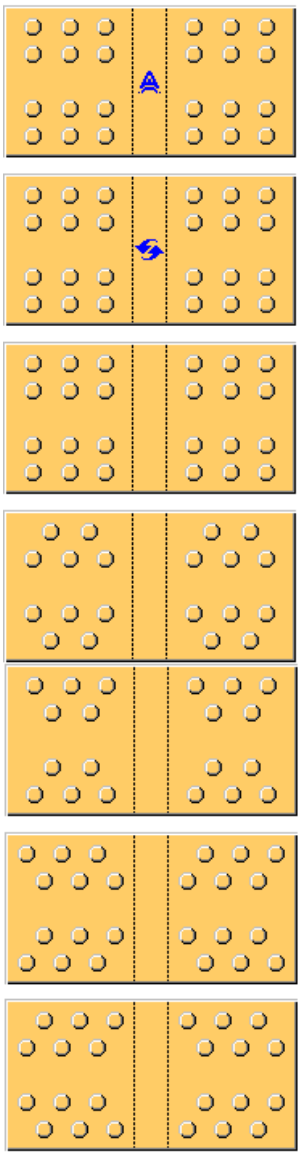
	Description
1	Dimension for vertical bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt offset



	Description
1	Define the bolt offset from the center line of the flange plate.

Staggering of bolts

Top flange bolts	Bottom flange bolts	Description
		Select the appropriate staggering option.

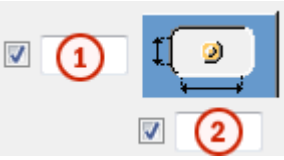
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.

Option	Description	Default
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.

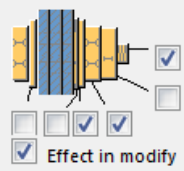


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Weld prep tab

Use the **Weld prep** tab to control weld backing bars and weld access holes. The weld backing bar is used behind the welds connecting the beam flanges.

Before you start

1. Set the thickness of flange plates to zero on the **Parts** tab. Flange welds and weld preparations are only created if the flange plates do not exist.
2. Click the **Weld** button and enter the correct weld size, type, angle, and other properties. Otherwise weld preparations are not created. Note that weld preparations are not created for all weld types.

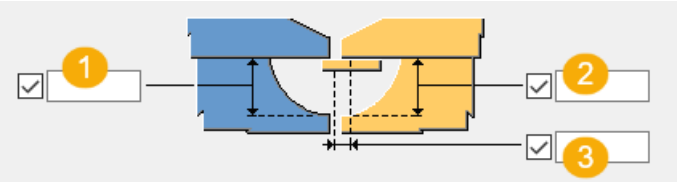
Backing bar

Option	Description
Backing bar	Backing bar thickness and width.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in

Option	Description	Default
		File menu --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

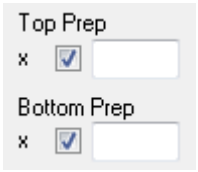
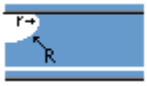
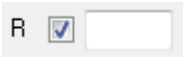
Weld access hole dimension



	Description
1	Define the weld access hole radius in the main part.
2	Define the weld access hole radius in the secondary part.
3	Define the weld backing bar offset in the direction of the secondary beam.

Weld access holes

Option	Description	Default
	Default Round weld access hole AutoDefaults can change this option.	
	Round weld access hole	
	Square weld access hole	
	Diagonal weld access hole	
	Round weld access hole with a radius that you can define in 	

Option	Description	Default
	Extended cone-shaped weld access hole with a radius and dimensions that you can define in and 	
	Cone-shaped weld access hole with radiuses that you can define in  and Capital **R** defines the large radius (height). Small **r** defines the small radius.	`R = 35` `r = 10`

Weld preps for flanges

Select whether the main or secondary part flange is beveled.

Weld preps for web

Select whether the main or secondary part web is beveled.

Include backing bar

Select whether the backing plate is welded to the main part, secondary part, none, or both.

Backing bar location

Select the backing bar location in relation to the flange.

Option	Description
Auto	Places the backing bars down for beams (**In-Out**) and inside (**In-In**) for columns.
In-In	Places both backing bars to the inner side of flanges.
In-Out	Places the top backing bars to the inner side of the top flange and the bottom back bar to the outer side of the bottom flange.

Weld backing bar length

Enter the length of the weld backing bar in the box below the options.

Option	Description
	Default Absolute length of the backing bar AutoDefaults can change this option.
	Absolute length of the backing bar
	Extension beyond the edge of the flange

Deck plates tab

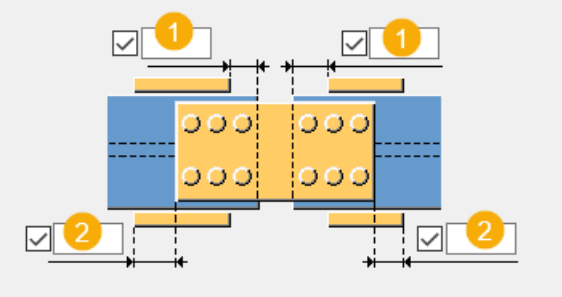
Use the **Deck plates** tab to define whether deck plates are created.

Part

Option	Description
Deck supports	Deck plate thickness and width.

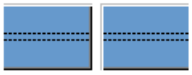
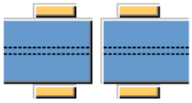
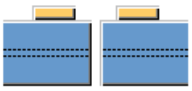
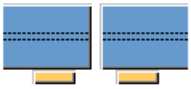
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Deck plate dimensions



1	Define the deck plate edge distance from the main part edge.
2	Define the deck plate dimension from the web plate edge.

Deck plate creation

Option	Description
	Default Deck plates are not created. AutoDefaults can change this option.
	Deck plates are not created.
	Deck plates are created on both sides.
	Deck plates are created on the front side.
	Deck plates are created on the back side.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Column splice (132)

Column splice (132) creates a splice connection between two columns.

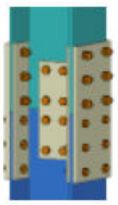
You can create the following splice connection types:

- Two outer flange plates with two web plates
- Four inner flange plates with two web plates
- Division plate with two outer flange plates and four web cleats
- Two outer flange plates, four inner flange plates, and two web plates

Objects created

- Web plate
- Flange plates
- Division plate
- Angle profile
- Web and flange shim plates
- Bolts
- Welds

Use for

Situation	Desciption
	Two columns connected with web and flange plates.

Limitations

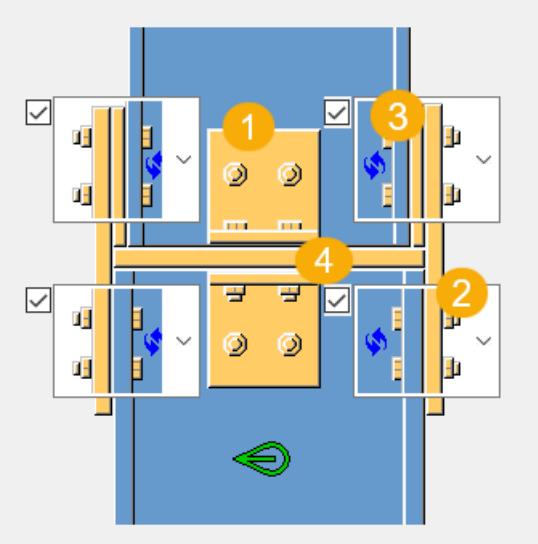
The columns must have a common center line or coinciding faces.

Selection order

1. Select the main part.
2. Select the secondary part.

   The connection is created automatically when you select the secondary part.

Part identification key



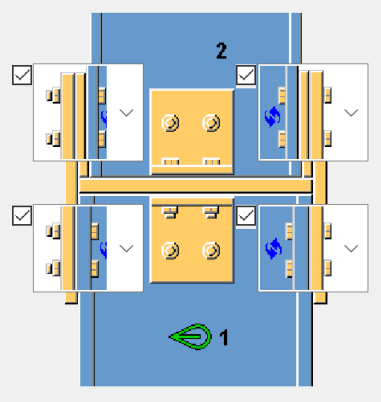
	Description
1	Web plate
2	Flange plate
3	Shim plate
4	Division plate

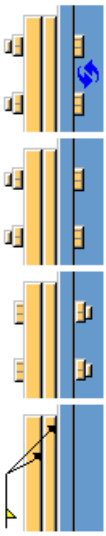
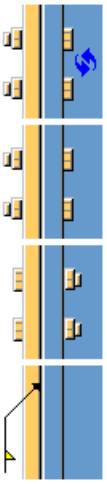
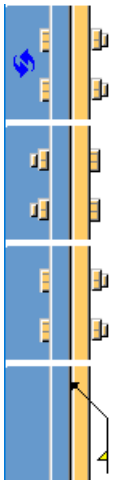
Picture tab

Use the **Picture** tab to define the bolting direction of the flange plates and to select whether the plates are bolted or welded.

Flange plate bolting

Select the bolting direction, or select to weld the plates to both the main part and the secondary part.



Option	Option	Option	Option
			

Parts tab

Use the **Parts** tab to define the part properties.

Parts

Option	Description	Default
Web plate	Thickness, width and height of the web plate.	
Ext flange plates	Thickness, width and height of the outer flange plates.	
Int flange plates	Thickness, width and height of the inner flange plates.	

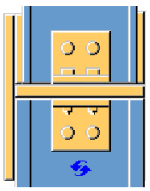
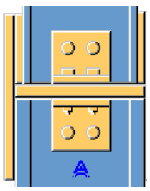
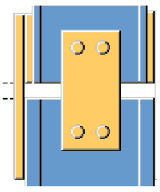
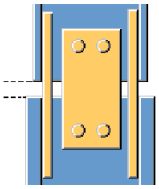
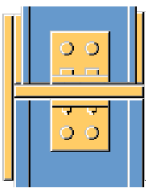
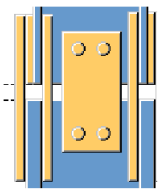
Option	Description	Default
Division plate	Thickness, width and height of the division plate.	The default thickness is 25 mm.
Angle profile	Select the angle profile from the profile catalog.	
Flange spacers	Thickness of the flange shim plates.	
Web spacers	Thickness of the web shim plates.	The default is 2 mm. If you enter a thickness, the largest number of plates of that thickness which fit in the gap will be created.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Parameters tab

Use the **Parameters** tab to define the flange plate and web plate edge cut type, tolerance between the columns, and angle profile and web plate creation.

Tolerance between columns

Option	Description
	Default No gap between the columns. Division plate is used. AutoDefaults can change this option.
	Automatic No gap between the columns. Division plate is used.
	Define the gap between the columns. Outer flange plate is created on one side.
	Define the gap between the columns. Inner flange plates are created.
	No gap between the columns. Division plate is used.
	Define the gap between the columns. Outer and inner flange plates are created.

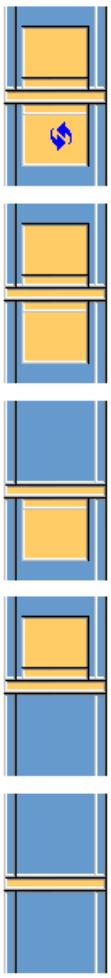
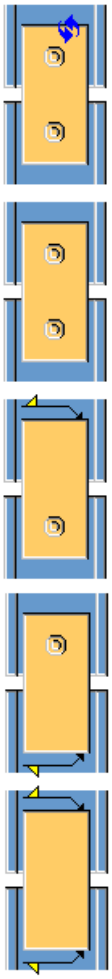
Angle profile attachment

Opton	Option	Description
		Default Angle profile is bolted. AutoDefaults can change this option.

Opton	Option	Description
		Angle profile is bolted.
		Angle profile is bolted and welded.
		Angle profile is bolted and welded.
		Angle profile is welded.

Web plate

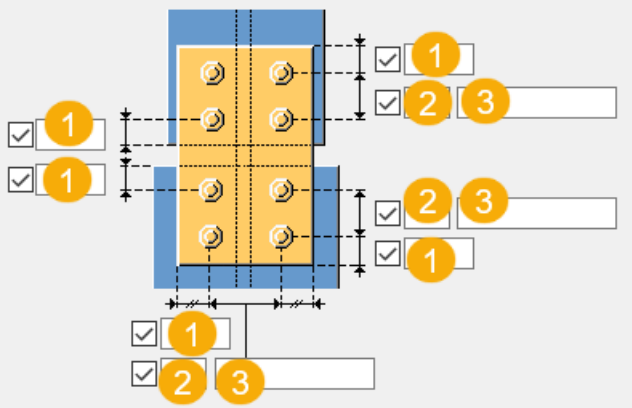
Select how the web plate is created. The options depend on whether a division plate or a gap is created between the columns.

Option	Option
	

Flange blt tab

Use the **Flange blt** tab to define the bolt group dimensions and bolt properties of flange bolts.

Bolt group dimensions



	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.
	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

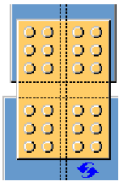
Workshop bolts

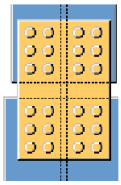
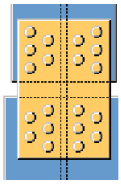
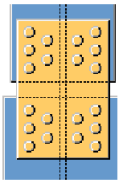
Select to which part the workshop bolts are attached.

- **None**: All bolts are site assembly.
- **Primary member**: Any bolts going to the main part are workshop bolts.
- **Secondary member**: Any bolts going to the secondary part are workshop bolts.

Staggering

Select an appropriate staggering option.

Option


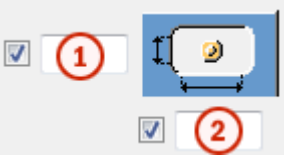
Option




Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.

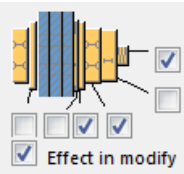


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

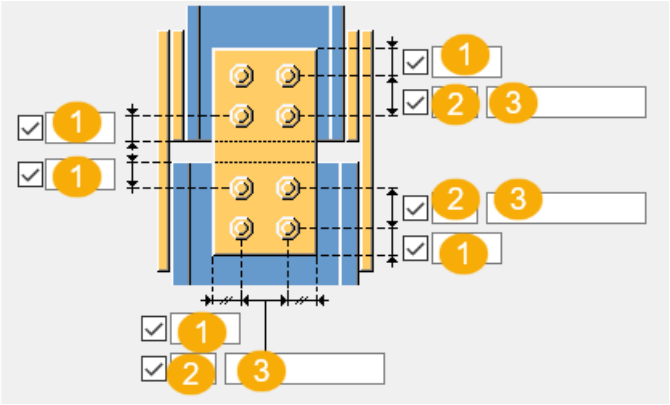
Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Web bolts tab

Use the **Web bolts** tab to define the bolt group dimensions and bolt properties of web bolts.

Bolt group dimensions



	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.

	Description
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

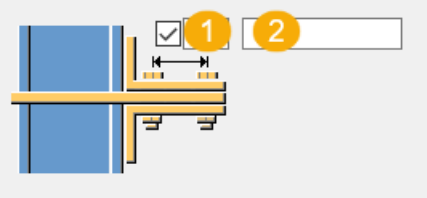
Workshop bolts

Select to which part the workshop bolts are attached.

- **None**: All bolts are site assembly.
- **Primary member**: Any bolts going to the main part are workshop bolts.
- **Secondary member**: Any bolts going to the secondary part are workshop bolts.

Angle bolts

Select where the angle bolts are attached: site or workshop.



	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

General tab

Click the link below to find out more:

Design tab

Click the link below to find out more:

If a division plate is used, the bearing is always **Yes**. The bearing is set to **No** only if the tolerance between the parts is > 0 mm.

Analysis tab

Click the link below to find out more:

Welds

Click the link below to find out more:

5.6 Welded connections

This section introduces welded connection components available in Tekla Structures.

- Cross (4) (page 1702)
- Offshore (9) (page 1706)
- Fitting (13) (page 1713)
- Round tube (23) (page 1717)
- Welded column (31) (page 1720)
- Welded tee (32) (page 1724)
- Weld preparation (44) (page 1738)
- New notch (49) (page 1742)
- Welded beam to beam (123) (page 1747)
- Welded column with stiffeners (128) (page 1756)
- Beam prep (183) (page 1776)
- Offshore (194) (page 1791)
- Cap plate detail (page 1794)
- Pipe doubler plate (page 1799)
- Ring plate (page 1806)

Cross (4)

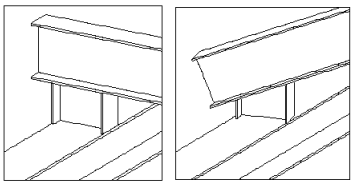
Cross (4) connects two beams that cross each other by using a profile welded to one of the beams. The beams should not collide with each other. Both horizontal and vertical angles between the profiles can vary.

Objects created

- Profile

- Welds

Use for

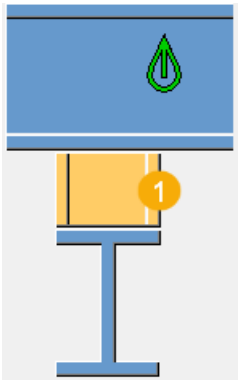
Situation	Description
	Two beams connected with a welded profile.

Selection order

1. Select the main part (beam).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

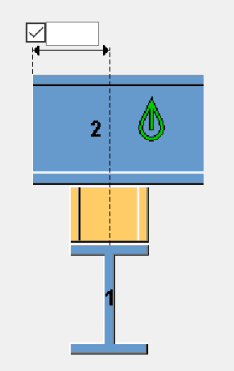


	Description
1	Profile

Picture tab

Use the **Picture** tab to define the connection dimensions.

Secondary part cut

Option	Description
	Define the secondary beam close end cut from the main beam center line. By default, the secondary beam is not cut.

Parts tab

Use the **Parts** tab to define the part properties.

Parts

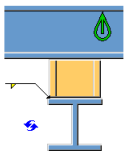
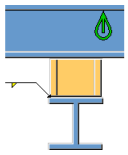
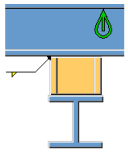
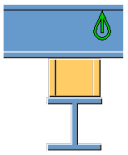
Option	Description
Profile	Select the profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

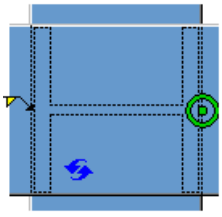
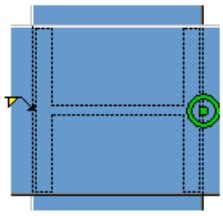
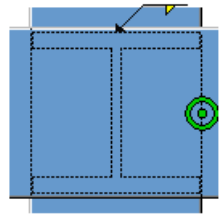
Parameters tab

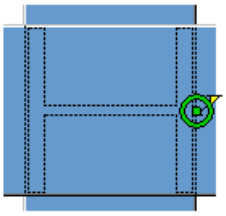
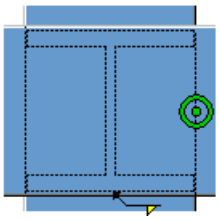
Use the **Parameters** tab to define the welding location and profile rotation.

Profile welded to

Option	Description
	Default Welded to the main part AutoDefaults can change this option.
	Welded to the main part
	Welded to the secondary part
	Not welded

Profile rotation

Option	Description
	Default Front AutoDefaults can change this option.
	Front
	Top

Option	Description
	Back
	Below

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Offshore (9)

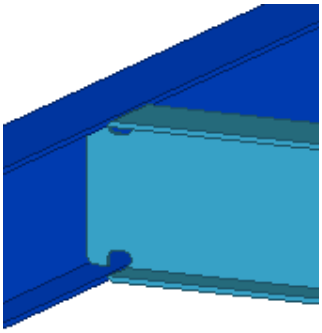
Offshore (9) connects a beam to another beam with welds. The component is designed to be used in offshore industry for creating notches and complex weld access holes, and to control the notch properties.

Objects created

- Notches

- Welds

Use for

Situation	Description
	Connection between two beams.

Profile limitations

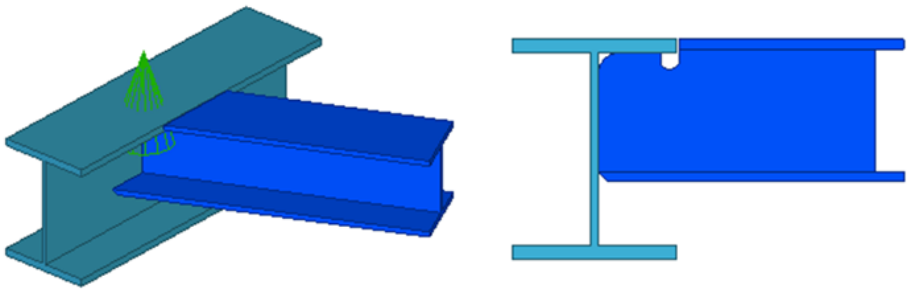
You can create three different notch types:

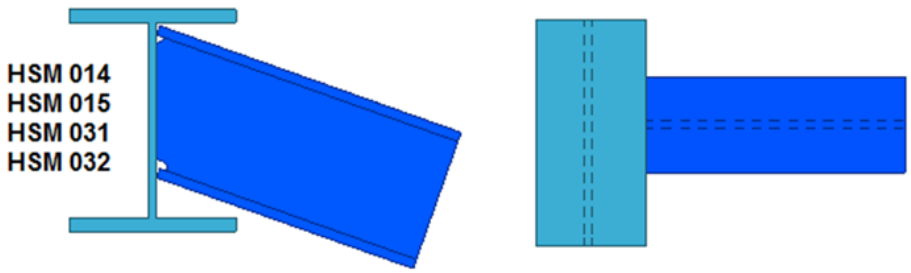
- **Free**
- **Heerema**
- **HSM**

When creating a **Free** notch type, the main and the secondary beam do not always have to have the same size. You can set them up according to your needs and create the connection.

When creating **Heerema** and **HSM** notch types, the beam size depends on the selected **Heerema** and **HSM** settings.

For all **Free** and **Heerema** types, and most **HSM** types, skewed secondary beams are allowed. Sloped secondary beams are only supported by some specific **HSM** types. See the table below for examples.

Notch type	Example
Free	

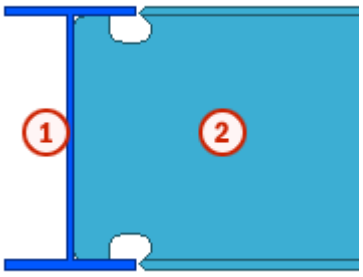
Notch type	Example
HSM	

Selection order

1. Select the main part (beam).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key



	Part
1	Beam
2	Beam

Picture 1 tab

Use the **Picture 1** tab to create notches in the direction of the web.

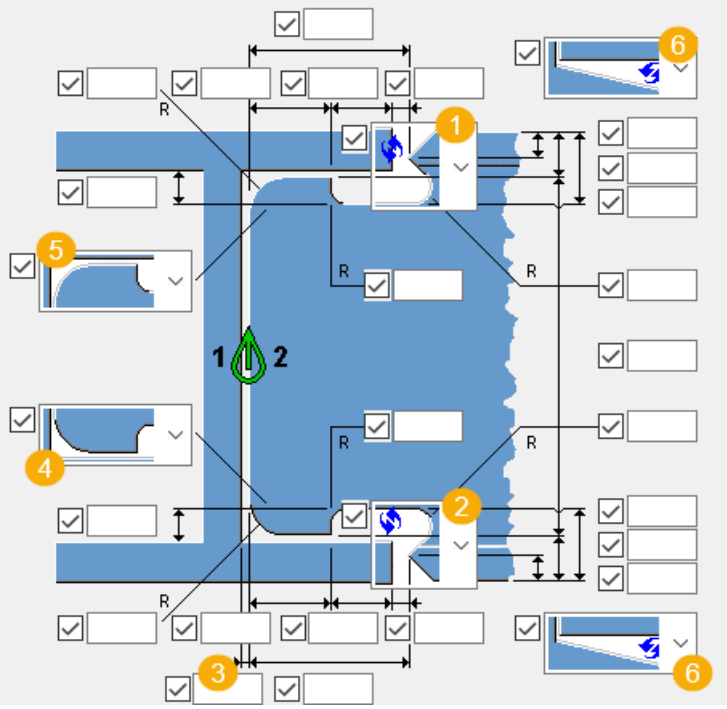
Notch type

Select the notch type and enter the notch values, if needed.

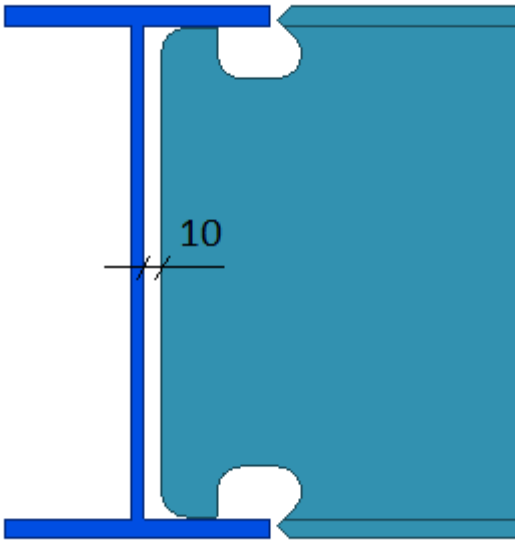
Option	Description
Free	Enter the notch dimensions.

Option	Description
Heerema	When you select this option, the predefined Heerema options are listed below. The predefined values are entered automatically in the user-defined attributes of the secondary beam.
HSM	When you select this option, the predefined HSM options are listed below. The predefined values are entered automatically in the user-defined attributes of the secondary beam.
No and clean up UDA	No notch is created. The user-defined attributes have no values.

Notches



	Description
1	Define the top flange notch for the secondary beam.
2	Define the bottom flange notch for the secondary beam.

	Description
3	Define a gap between the main beam web and the secondary beam end. 
4	Define the bottom rounding and a possible weld access hole for the secondary beam.
5	Define the top rounding and a possible weld access hole for the secondary beam.
6	Create a cut that follows the sloped flange of the secondary beam.

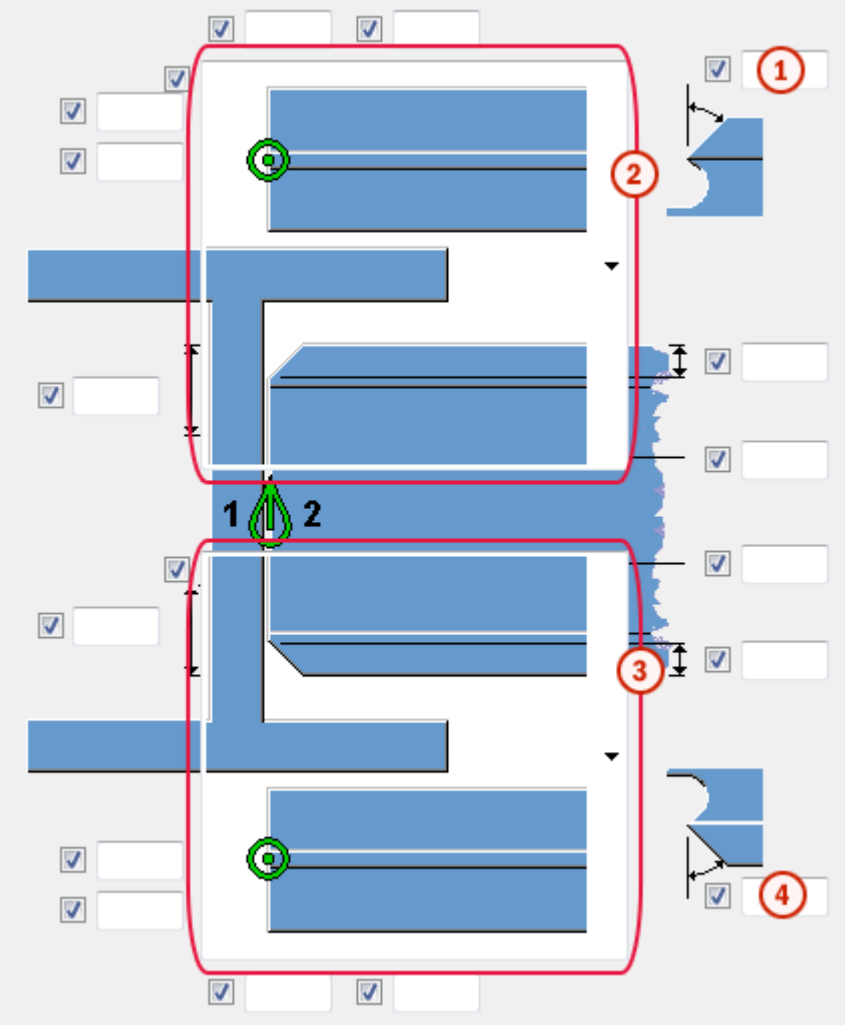
Fitting

Select the type of the fitting.

Picture 2 tab

Use the **Picture 2** tab to create notches in the direction of the flange. The options on this tab can only be used if the notch type is set to **Free** on the **Picture 1** tab. If you select a special type of notch (Heerema, HSM) on the **Picture 1** tab, then the predefined values are entered automatically in the user-defined attributes.

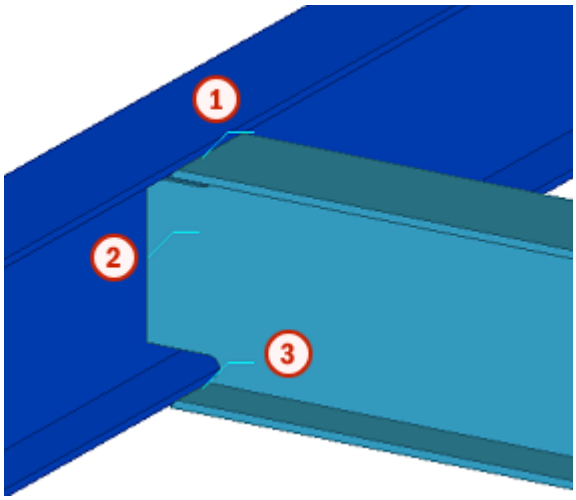
Notches



	Description
1	Define the bevel angle for the top flange (optional).
2	Define the top flange bevel.
3	Define the bottom flange bevel.
4	Define the bevel angle for the bottom flange (optional).

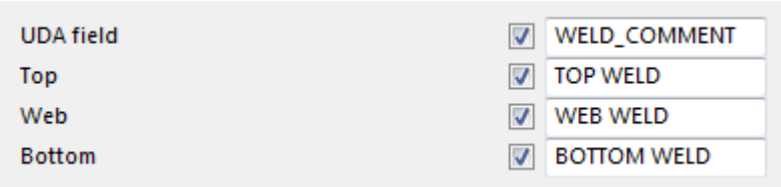
Weld description tab

Use the **Weld description** tab to control weld descriptions for the top, web and bottom welds.

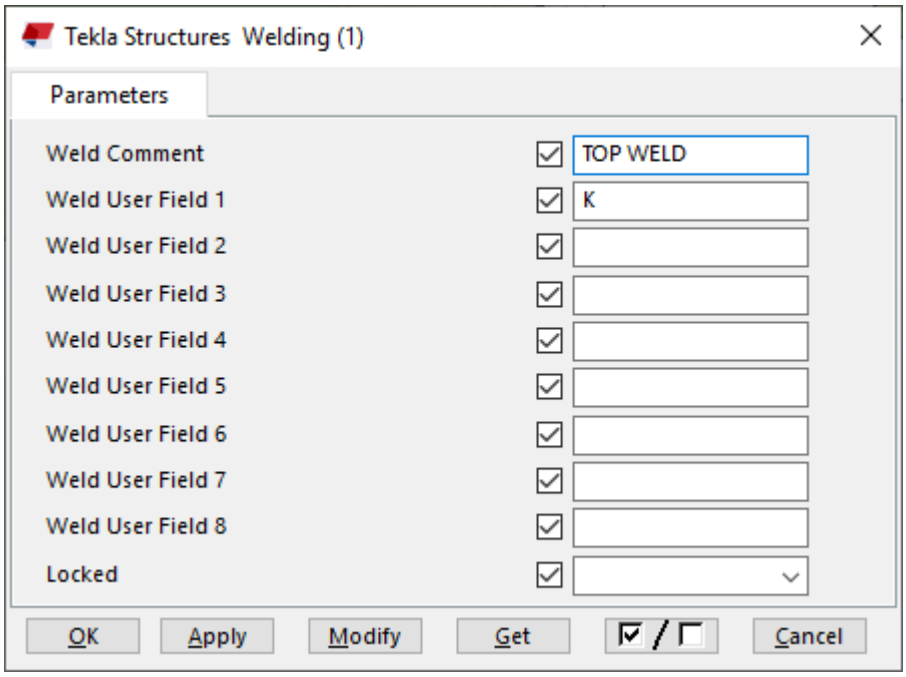


	Description
1	Top weld
2	Web weld
3	Bottom weld

Use the boxes on the **Weld description** tab to define a name for the weld comment. Enter the text `WELD_COMMENT` in the **UDA field**. The descriptions in the **Top**, **Web** and **Bottom** boxes correspond to each weld.



Double-click a weld in the model and go to the user-defined attributes of the weld. The text is entered in the **Weld Comment** box.



General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

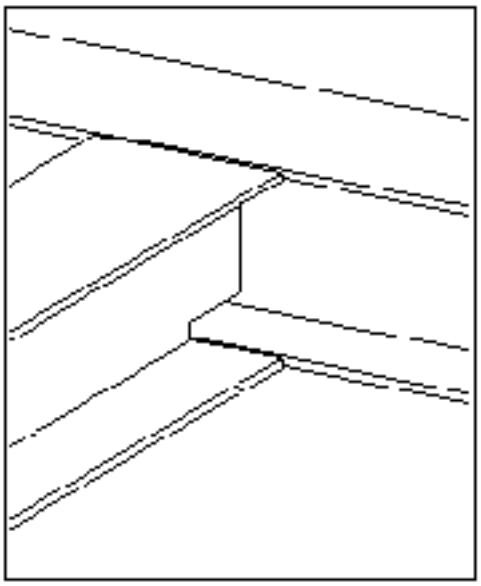
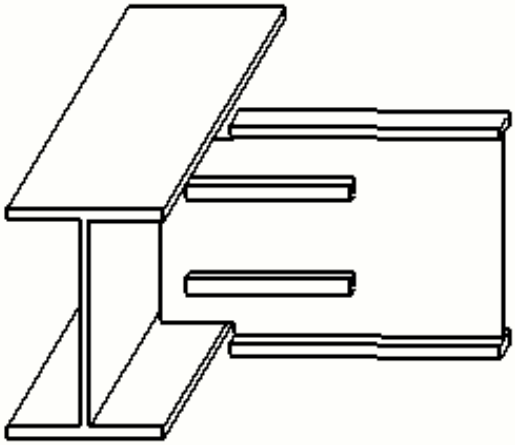
Fitting (13)

Fitting (13) connects a beam to a beam using welds.

Objects created

- Stiffeners (2) (optional)
- Welds

Use for

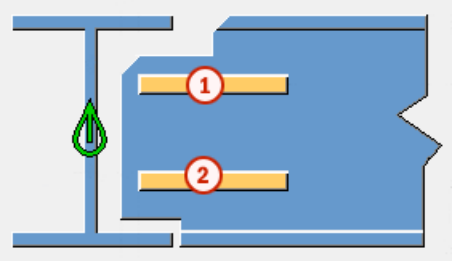
Situation	Description
	Fitting connection without stiffeners.
	Fitting connection with upper and lower stiffener.

Selection order

1. Select the main part (beam).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary beam is selected.

Part identification key

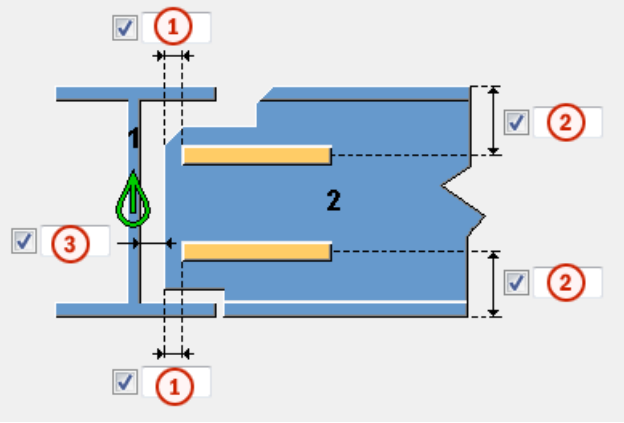


	Part
1	Upper stiffener
2	Lower stiffener

Picture tab

Use the **Picture** tab to control the fitting dimensions.

Dimensions



	Description
1	Stiffener horizontal distance from the secondary beam edge.
2	Stiffener vertical distance from the secondary beam edge.
3	Distance between the main beam and the secondary beam.

Option	Description
Max gap allowed to end plate	Enter the tolerance value for fitting the secondary part. Use this option when the secondary part is skewed. The gap is calculated

Option	Description
	using the skew angle and the height of the secondary profile. If the gap is within the tolerance value, the secondary part is fitted in the main part coordinate system. When the defined maximum gap is greater than the actual gap in the model, the secondary part is fitted in the secondary part coordinate system.
Weldings	Select whether welds are created. Welds are created when you select **Yes**. The default is that welds are not created.

Parts tab

Use the **Parts** tab to control the stiffener properties.

Stiffener

Part	Description
Upper stiffeners	Thickness, width, and height of the upper stiffener.
Lower stiffeners	Thickness, width, and height of the lower stiffener.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

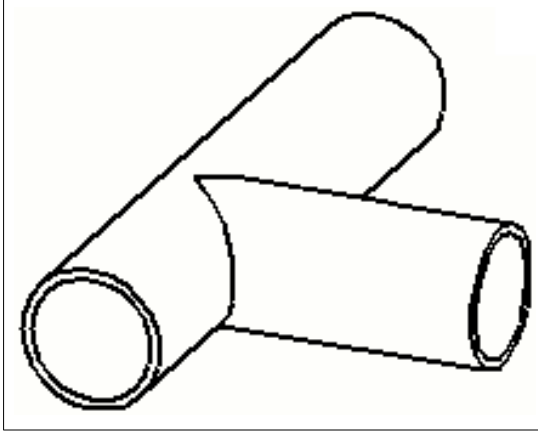
Round tube (23)

Round tube (23) connects two round tube profiles with a fitting and welds.

Objects created

- Fitting
- Welds

Use for

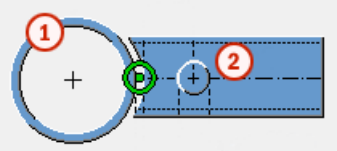
Situation	Description
	Connection between two round tubes. Fitting and welds are created.

Selection order

1. Select the main part (first round tube).
2. Select the secondary part (second round tube).

   The connection is created automatically when the secondary part is selected.

Part identification key

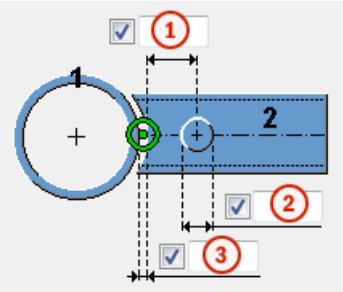


	Part
1	Round tube
2	Round tube

Picture tab

Use the **Picture** tab to control the dimensions of the secondary round tube fitting and the gap between the round tubes.

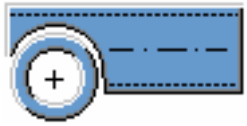
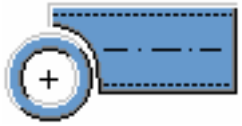
Part dimensions



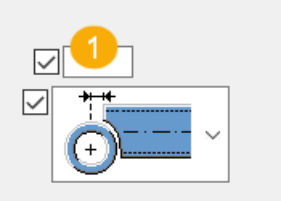
	Description	Default
1	Pop mark offset. You can define the offset if you have set the **Position of pop mark** option to **At minimum angle**, **At maximum angle**, or **Both**.	`100 mm`
2	Pop mark diameter. You can define the diameter if you have set the **Position of pop mark** option to **At minimum angle**, **At maximum angle**, or **Both**.	`10 mm`

	Description	Default
3	Gap between the main and the secondary round tube.	

Secondary part fitting

Option	Description
	Default The secondary round tube is cut with the main round tube. AutoDefaults can change this option.
	The secondary round tube is cut with the main round tube.
	The secondary round tube is fitted to the center line of the main round tube and then cut.

Secondary part cut



	Description
1	Define the secondary part cut distance from the main part center.

Position of pop mark

Select the position of the pop mark.

Parameters tab

Use the **Parameters** tab to control whether welds are created and to define the rotation angle and cut tolerance.

Option	Description
Weldings	Define whether welds are created between the round tubes.

Option	Description
Rotation angle of cut part	Define the rotation angle of the cut in the main round tube. In some cases it is necessary to rotate the cut part in the main round tube to make sure the cut is shown correctly. The default is that the cut part is not rotated.
Cut tolerance of sec	Define the cut tolerance of the secondary round tube.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the links below to find out more:

Analysis tab

Welds

Click the link below to find out more:

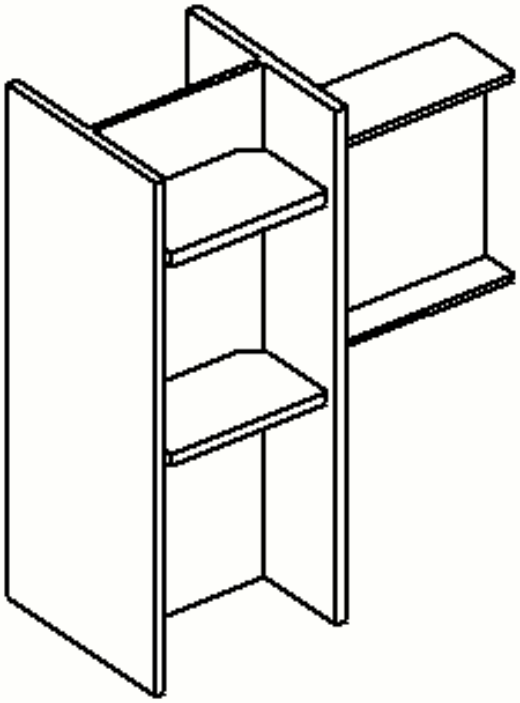
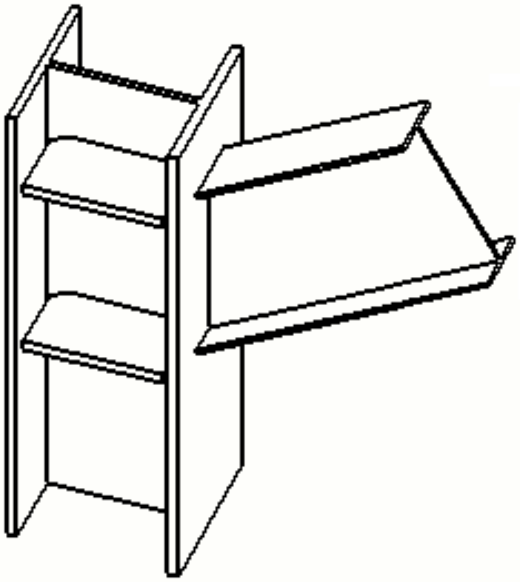
Welded column (31)

Welded column (31) connects a beam to a column flange using fittings and welds. Stiffeners are created between the column flanges.

Objects created

- Stiffeners
- Fittings
- Welds

Use for

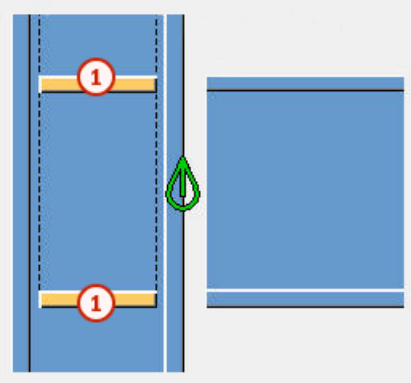
Situation	Description
	Beam welded to a column. Four stiffeners are created.
	Sloped beam welded to a column. Four stiffeners are created.

Selection order

1. Select the main part (column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

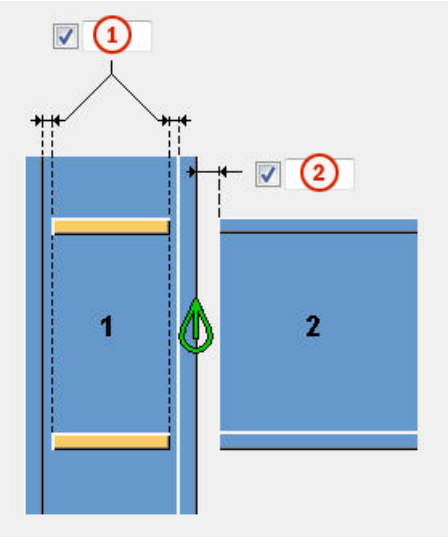


	Part
1	Stiffener

Picture tab

Use the **Picture** tab to control the dimensions and stiffener positions.

Dimensions



	Description
1	Distance between the stiffener and the column flange.
2	Distance between the column flange and the edge of the beam. You can define the distance if the **Fit secondary** option is set to **Default** or **Yes**.

Fit secondary

Define whether the secondary beam can be fitted to the column.

Parts tab

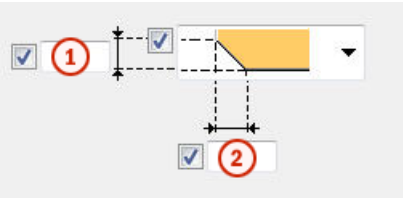
Use the **Parts** tab to control the stiffener properties.

Stiffeners

Part	Description	Default
Stiffeners	Thickness, width, and height of the stiffeners.	Thickness = `16 mm` Width = If the width is not defined, it is based on the flange width. Height = Column web height

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Chamfer dimensions



	Description
1	Vertical chamfer dimension. You can define the vertical dimension for line chamfers.
2	Horizontal chamfer dimension.

Chamfer type

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

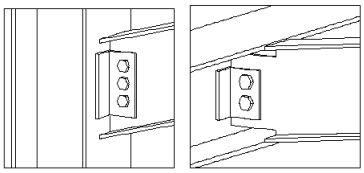
Welded tee (32)

Welded tee (32) connects a beam to a column or to another beam using a T-shaped piece of an I-profile welded to the main part and bolted to the secondary beam web.

Objects created

- Tee part
- Bolts
- Welds

Use for

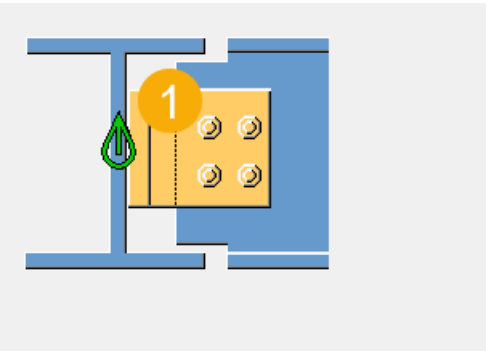
Situation	Description
	Welded tee connected to a column and bolted to a beam.

Selection order

1. Select the main part.
2. Select the secondary part.

   The connection is created automatically when the secondary part is selected.

Part identification key

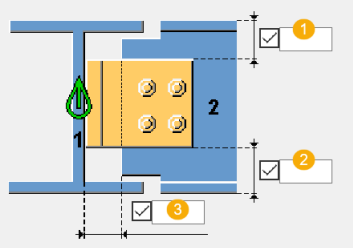


	Description
1	Tee part

Picture tab

Use the **Picture** tab to define the connection dimensions.

Dimensions



	Description	Default
1	Tee part edge distance from the secondary part top.	
2	Tee part edge distance from the secondary part bottom.	
3	Gap between the main part and the secondary part. The value you define affects the tee part size.	`10 mm`

Parts tab

Use the **Parts** tab to define the part properties.

Parts

Option	Description	Default
Profile	Select the profile from the profile catalog.	The same as the secondary part profile.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Option	Description	Default
Finish	Describes how the part surface has been treated.	

Parameters tab

Use the **Parameters** tab to define the tee part position.

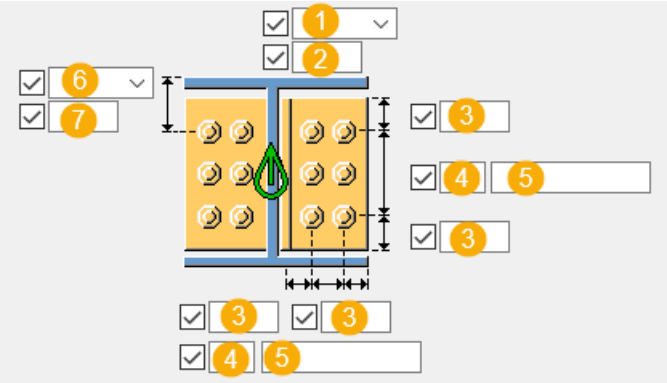
Tee part position

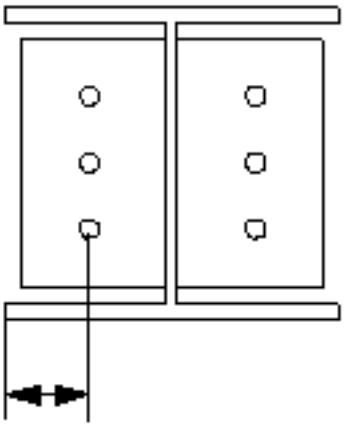
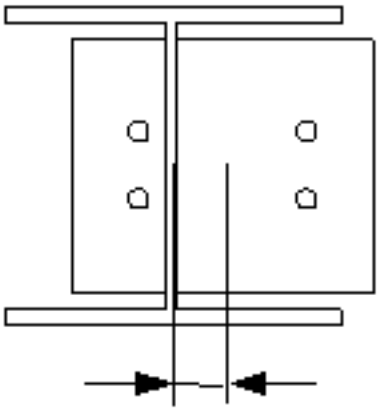
Option	Description
	Default Tee part is on the left side. AutoDefaults can change this option.
	Tee part is on the left side.
	Tee part is on the right side.

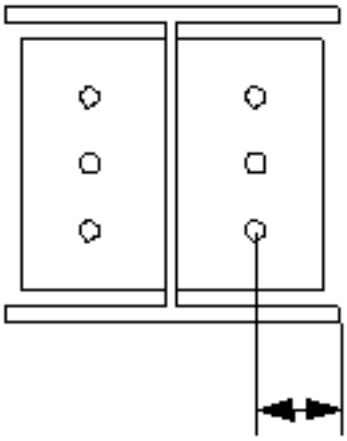
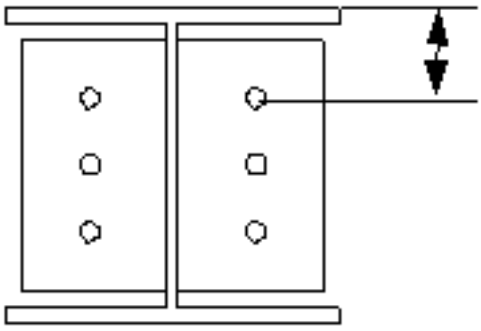
Primary bolts tab

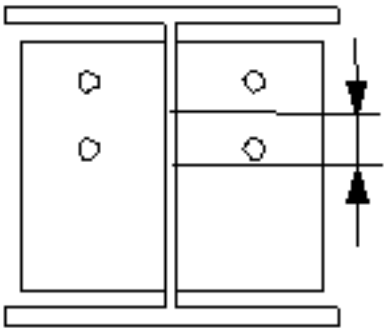
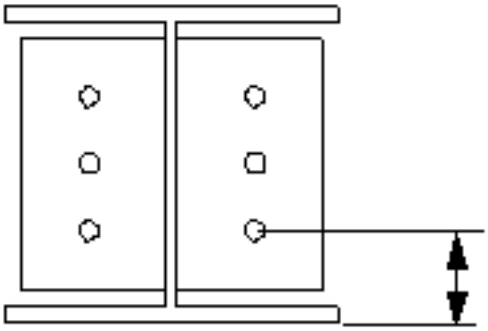
Use the **Primary bolts** tab to define the main part bolt group dimensions and bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt.  • **Middle**: From the center line of the secondary part to the center line of the bolts. 

	Description
	• **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Dimension for vertical bolt group position.

Connect to primary

Select how the profile is connected to the main part.

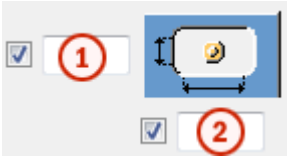
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within	`Yes`

Option	Description	Default
	the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

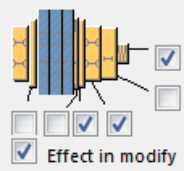


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

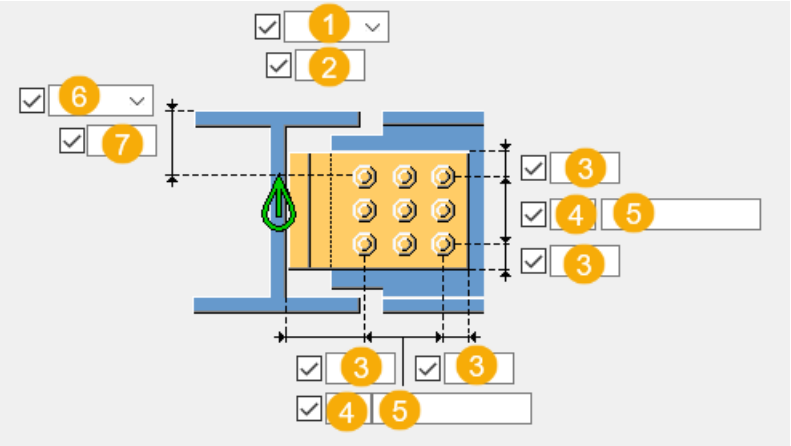
Staggering of bolts

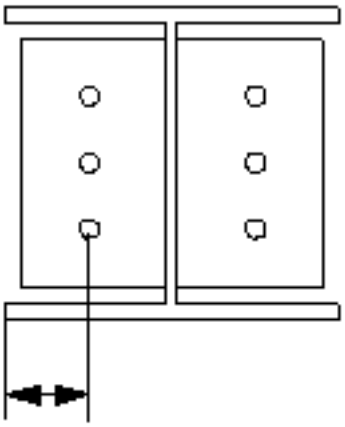
Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

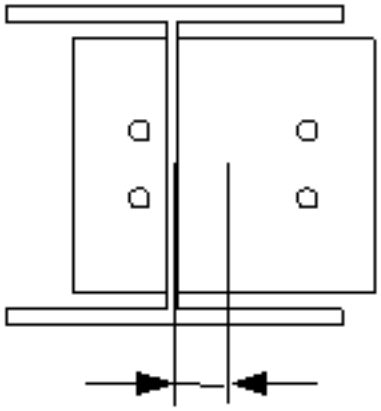
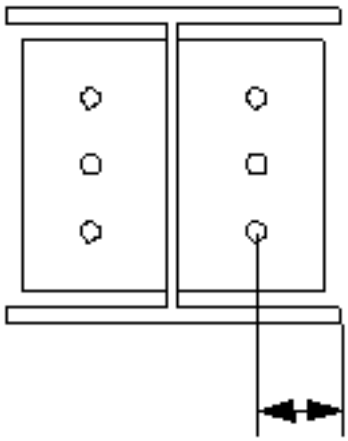
Secondary bolts tab

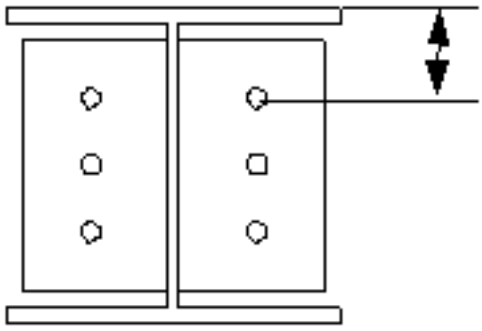
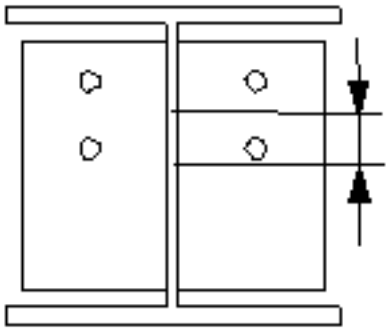
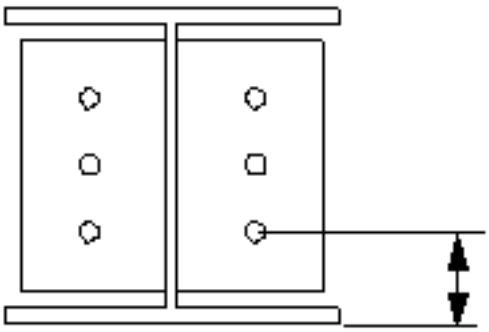
Use the **Secondary bolts** tab to define the secondary part bolt group dimensions and bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt. 

	Description
	• **Middle**: From the center line of the secondary part to the center line of the bolts.  • **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

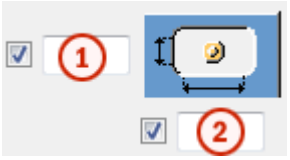
	Description
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Dimension for vertical bolt group position.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



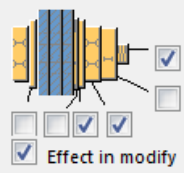
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	

Option	Description	Default
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3

Option	Description
	Staggered type 4

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

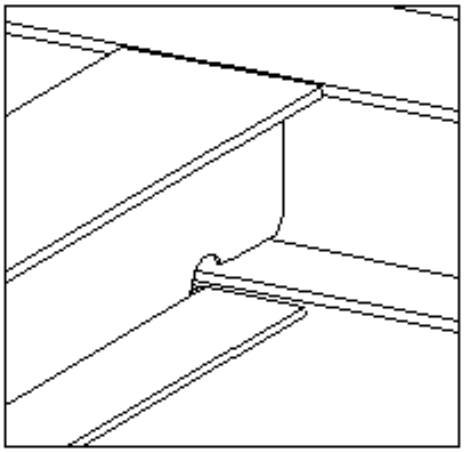
Weld preparation (44)

Connects two beams of I profile with welds. You can shape the secondary beam end with rat holes and other weld preparations. The only allowed beam profile is I-profile.

Objects created

- Welds
- Fitting
- Cuts to shape the beam end

Use for

Situation	Description
	Beam welded to a beam web. Rat hole at the bottom.

Before you start

Create two beams.

Selection order

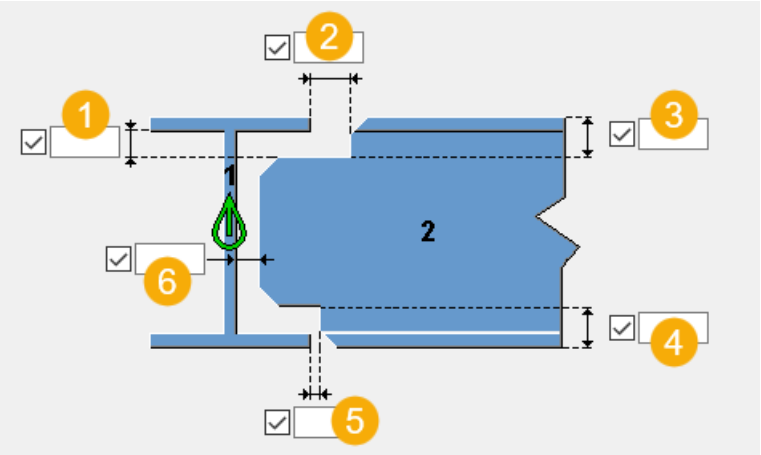
1. Select the main part.
2. Select the secondary part.

   The connection is created automatically when the secondary part is selected.

Picture tab

Use the **Picture** tab to define the connection geometry.

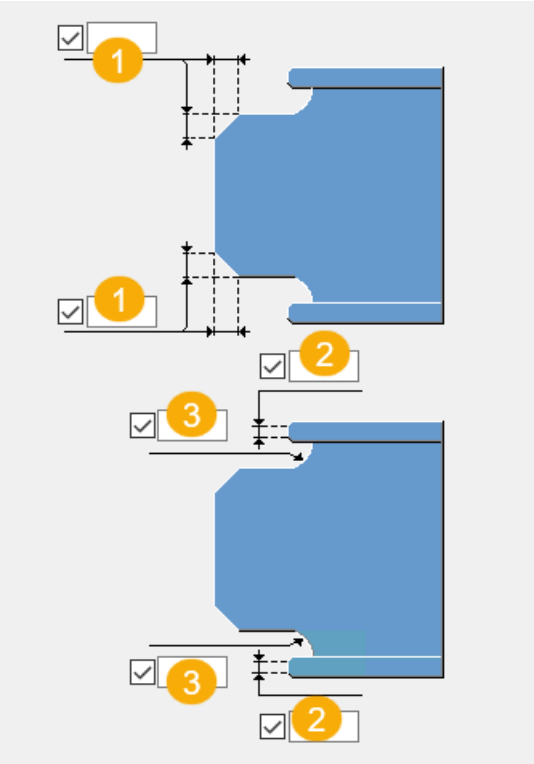
Dimensions



	Description	Default
1	Vertical tolerance. Tolerance between the main part web and the secondary part flange.	0.2*secondary part flange thickness
2	Flange horizontal tolerance. Horizontal tolerance between the secondary part and the main part upper flange.	3 mm
3	Web upper vertical tolerance. Vertical distance between the secondary part upper edge and the top of the web.	
4	Web lower vertical tolerance. Vertical distance between the secondary part lower edge and the bottom of the web.	
5	Flange horizontal tolerance Horizontal tolerance between the secondary part and the main part lower flange.	3 mm
6	Web horizontal tolerance. Tolerance between the webs of the beams.	0 mm

Parameters tab

Beam end shape



	Description
1	Web chamfer Dimensions of the top and bottom chamfers in the secondary part web.
2	Top and bottom flange straight distance.
3	Top and bottom weld access hole radius.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

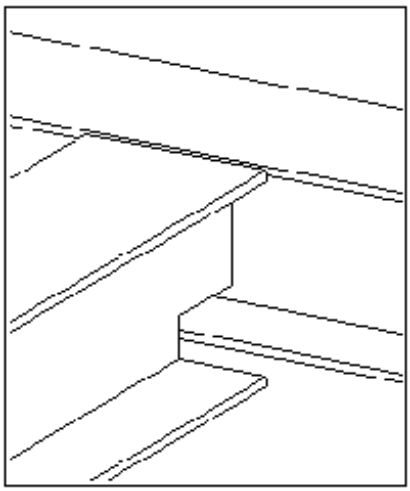
New notch (49)

New notch (49) welds a beam to another beam. You can define how to cut the beam end. The connection also creates horizontal stiffeners, if needed.

Objects created

- Cuts to shape beam end
- Welds
- Stiffeners (4) optional

Use for

Situation	Description
	A beam welded to a beam. Bottom of the secondary beam web notched.

Before you start

Create two beams.

Selection order

1. Select the main part (beam).

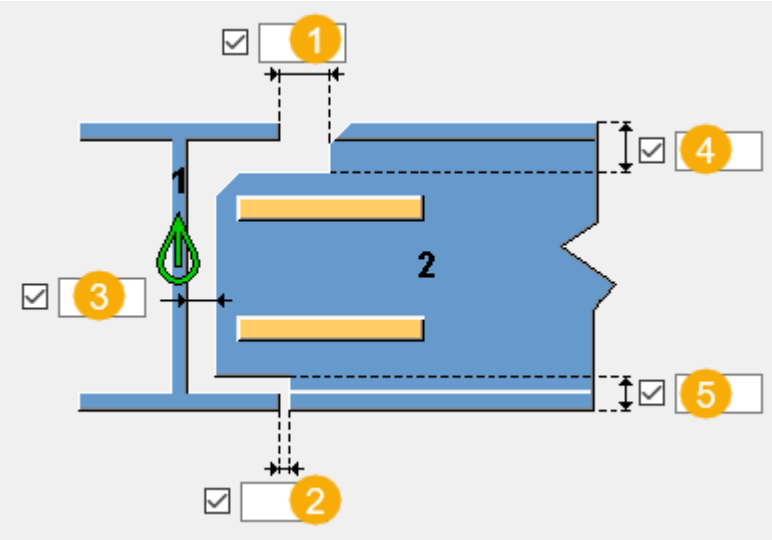
2. Select the secondary part (beam to be cut).

   The connection is created automatically when the secondary part is selected.

Picture tab

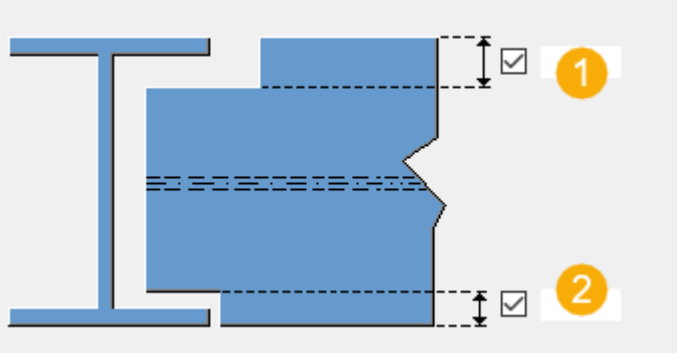
Use the **Picture** tab to define the connection geometry.

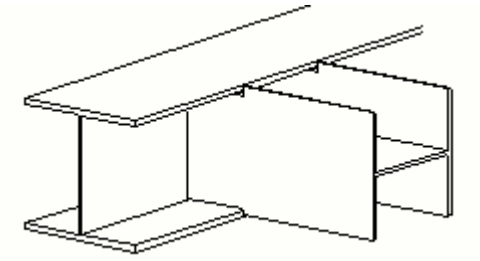
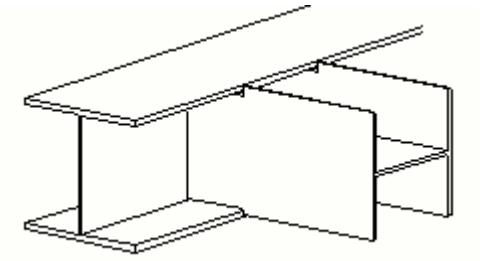
Dimensions



	Description	Default
1	Horizontal tolerance between the main part and the secondary part upper flanges.	0 mm
2	Lower flange horizontal tolerance between the main part and the secondary part lower flanges.	0 mm
3	Web horizontal tolerance between the main part web and the secondary part web.	0 mm
4	Web upper vertical tolerance. Vertical distance between the secondary part upper edge and the top of the web.	
5	Web lower vertical tolerance. Vertical distance between the secondary part lower edge and the bottom of the web.	

Secondary beam flange notch dimensions



	Description
1	Notch dimension of the secondary beam top or bottom flange, if the secondary beam is rotated as in the picture below. 
2	Notch dimension of the secondary beam top or bottom flange, if the secondary beam is rotated as in the picture below. 

Parts tab

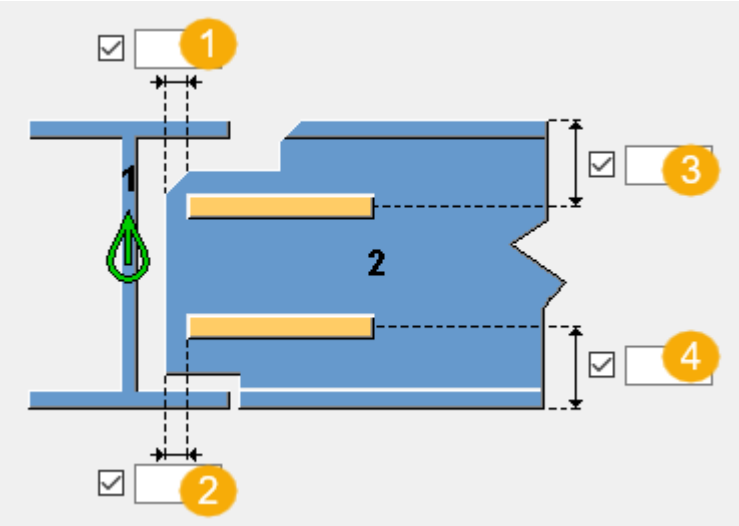
Option	Description	Default
Upper/lower stiffeners	Stiffener thickness, width and height.	The default thickness is equal to the beam's upper / lower flange thickness. The default width is 0.5*(upper / lower

Option	Description	Default
		flange width-web thickness). The default length (height) is 300 mm.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

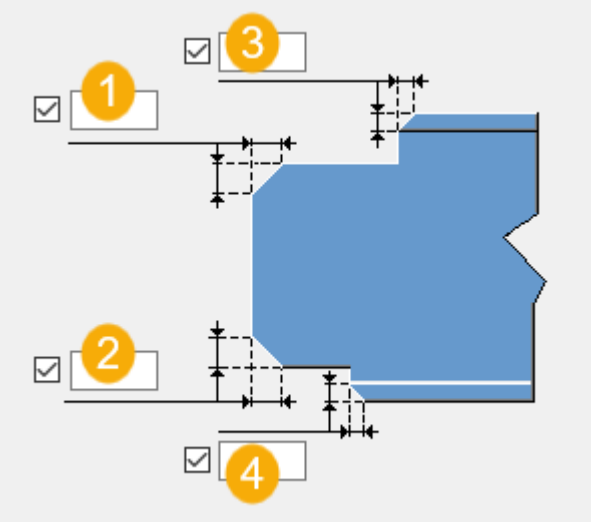
Parameters tab

Stiffener position dimensions



	Description	Default
1	Upper stiffener horizontal distance from the beam end.	0 mm
2	Lower stiffener horizontal distance from the beam end.	0 mm
3	Upper stiffener vertical distance from the upper flange.	Secondary beam height / 4
4	Lower stiffener vertical distance from the lower flange.	Secondary beam height / 4

Secondary beam chamfer dimensions



	Description	Default
1	Upper web chamfer dimension.	0 mm
2	Lower web chamfer dimension.	0 mm
3	Upper flange chamfer dimension.	0 mm
4	Lower flange chamfer dimension.	0 mm

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Welded beam to beam (123)

Welded beam to beam (123) connects two beams using a fully welded connection. The beams can have welding preparations. Stiffener plates with welding preparations can also be created.

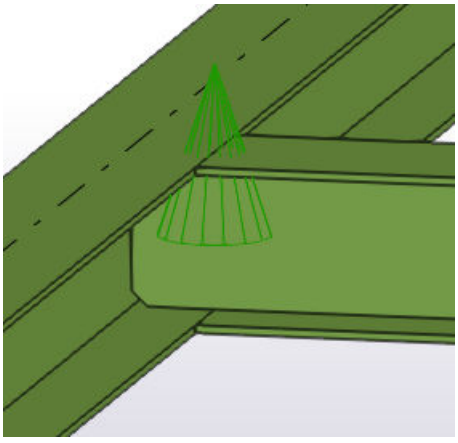
Limitations

- You can only use I-section beams.
- You cannot use beams that are horizontally skewed or vertically sloped.

Objects created

- Stiffeners
- Cuts
- Welds

Use for

Situation	Description
	Fully welded beam to beam connection

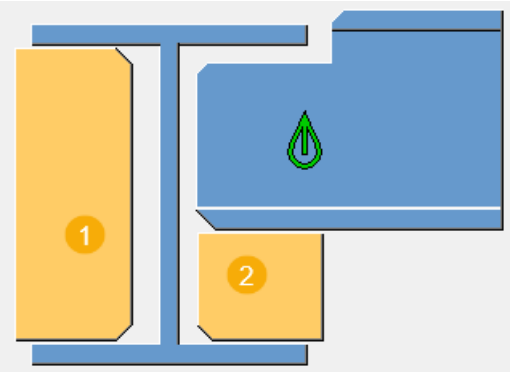
Selection order

1. Select the main part (beam).

2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

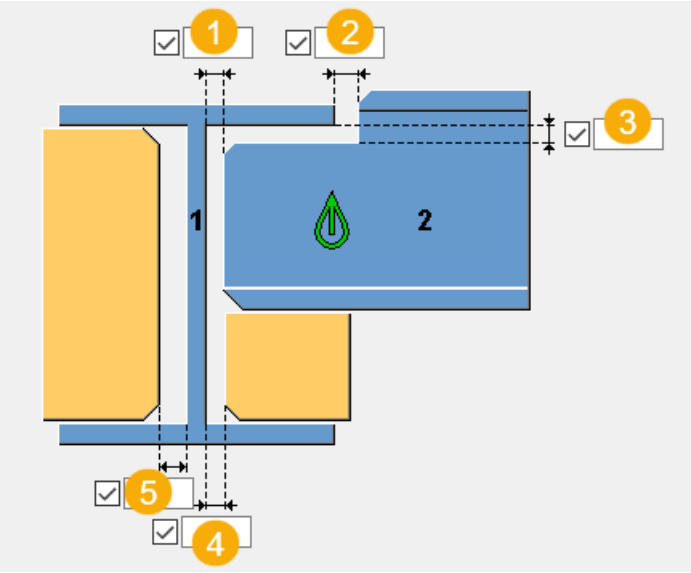


	Description
1	Front stiffener
2	Back stiffener

Picture tab

Use the **Picture** tab to define the beam and stiffener offset dimensions.

Dimensions



	Description
1	Gap dimension between the main part web and the secondary part edge.
2	Secondary part offset dimension from the main part flange.
3	Secondary part offset dimension from the underside of the main part flange.
4	Back stiffener offset dimension from the main part web.
5	Front stiffener offset dimension from the main part web.

Parameters tab

Use the **Parameters** tab to define the stiffener plate properties, and whether weld preparations are required.

Plates

Option	Description
Btm stiff plate	Thickness and width of the stiffener on the main beam side.
Back stiffener	Thickness and width of the stiffener located behind the secondary beam.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Weld preparation creation, beam web and flange thickness

Option	Description	Default
Are weld preparations required	Select whether to create weld preparations.	Weld preparations are not created.

Option	Description	Default
Root face thickness of web at U/S of flange	Set the minimum thickness of the main beam web after the weld preparation cut. The minimum value is `2.0` mm. If you do not enter a value, the minimum value is used. Note that if you have defined a double-sided weld preparation, the root face thickness is centred on the web of the beam.	
Root face thickness of web	Set the minimum thickness of the incoming beam web after the weld preparation cut. The minimum value is `2.0` mm. Note that if you have defined a double-sided weld preparation, the root face thickness is centred on the web of the beam.	
Root face thickness for front stiffener	Set the minimum thickness of the front stiffener after the weld preparation cut. The minimum value is `2.0` mm. Note that if you have defined a double-sided weld preparation, the root face thickness is centred on the stiffener.	The default value is the thickness of the flange, that is, no weld preparation other than a gap can be created unless you define a root face thickness.
Root face thickness for rear stiffener	Set the minimum thickness of the back stiffener after the weld preparation cut. The	The default value is the thickness of the flange, that is, no weld preparation other than a gap can be created

Option	Description	Default
	minimum value is `2.0` mm. Note that if you have defined a double-sided weld preparation, the root face thickness is centred on the stiffener.	unless you define a root face thickness.
Use flange slope ratio	Select whether the flange slope ratio is used in notch cuts.	

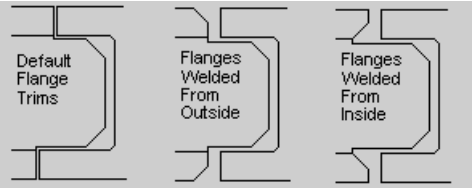
Define weld preparations

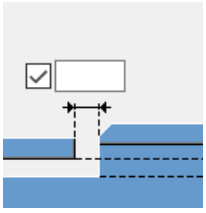
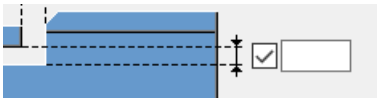
You can apply various weld preparations to the main beam flanges and web, and to the stiffener plates.

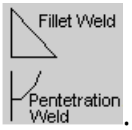
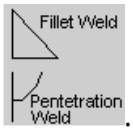
By default, the connection automatically detects whether the top and bottom flanges of the main beam clash with the flanges of the secondary beam. This determines how the weld preparations are handled.

If the main beam does not have the same section size as the secondary beam, the connection detects which flange is free based on which of the top or the bottom flanges are still clashing.

The following explains the basic weld preparation situations of each section of the main beam:

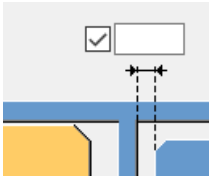


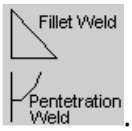
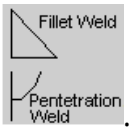
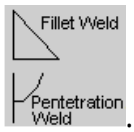
Situation	Description
Default	The top flange is cut back to avoid the top flange of the secondary beam. You can define the secondary part offset from the main part flange in  and  on the **Picture** tab. Set **Are weld preparations required** on the **Parameters** tab to **Yes**.

Situation	Description
Weld from the outside	To create a weld preparation that shapes the top flange so that it can be welded from the outside, first set **Are weld preparations required** on the **Parameters** tab to **Yes**. Click the **Welds** button to define the weld settings: • For top flange: For weld number **1**, change the top half of **Type** from fillet weld to penetration weld  . You can also define the size and angle of the weld. • For the bottom flange: Modify weld number **3**.
Weld from the inside	To create a weld preparation that shapes the top flange so that it can be welded from the inside, first set **Are weld preparations required** on the **Parameters** tab to **Yes**. Click the **Welds** button to define the weld settings: • For weld number **1**, change the bottom half of **Type** from fillet weld to penetration weld  . You can also define the size and angle of the weld. • For the bottom flange: Modify weld number **3**.

Main beam web

Note that if you define any weld preparation on the beam web, the weld preparation is created on the vertical face of the web and the horizontal face of the web at the underside of the secondary beam flange.

Situation	Description
Default situation	The default weld preparation of the main beam web is to fit the beam to the side of the secondary beam web. Set **Are weld preparations required** on the **Parameters** tab to **Yes**. If needed, define that a gap is created between the two webs in  on the **Picture** tab.

Situation	Description
Weld from the near side	Set **Are weld preparations required** on the **Parameters** tab to **Yes**. Click the **Welds** button to define the weld settings. For weld number **2**, change the top half of **Type** from fillet weld to penetration weld  . You can also define the size and angle of the weld. Check on the **Parameters** tab that the root face thickness values are as needed.
Weld from the far side	Set **Are weld preparations required** on the **Parameters** tab to **Yes**. Click the **Welds** button to define the weld settings. For weld number **2**, change the bottom half of **Type** from fillet weld to penetration weld  . You can also define the size and angle of the weld. Check on the **Parameters** tab that the root face thickness values are as needed.
Weld from both sides	Set **Are weld preparations required** on the **Parameters** tab to **Yes**. Click the **Welds** button to define the weld settings. For weld number **2**, change both the top and the bottom halves of **Type** from fillet weld to penetration weld  . You can also define the size and angle of the weld. Check on the **Parameters** tab that the root face thickness values are as needed.

Front and back stiffener plates

If you define any weld preparation on the front and back stiffener, it is created on the three stiffener sides that are welded to the main beam and the secondary beam.

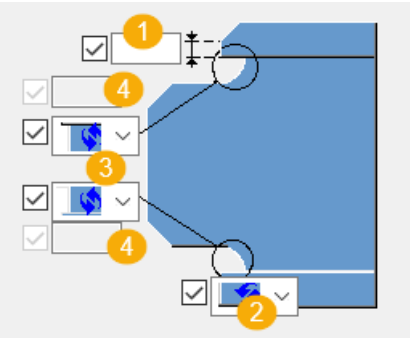
Situation	Description
Default situation	For the default situation, see the description of the main beam web. The welds you need to define are weld

Situation	Description
	number **5** for the front stiffener and weld number **6** for the back stiffener.
Weld from the near side	See the description of the main beam web. The welds you need to define are weld number **5** for the front stiffener and weld number **6** for the back stiffener.
Weld from the far side	See description of the main beam web. The welds you need to define are weld number **5** for the front stiffener and weld number **6** for the back stiffener.
Weld from both sides	See the description of the main beam web. The welds you need to define are weld number **5** for the front stiffener and weld number **6** for the back stiffener.

Chamfers tab

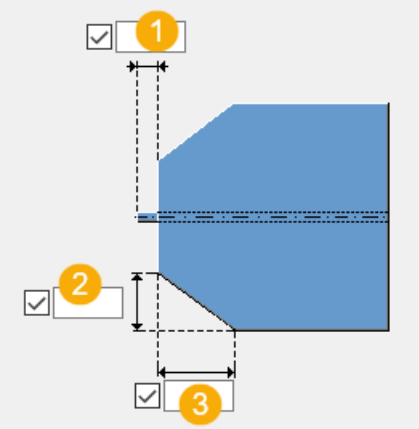
Use the **Chamfers** tab to define the weld access hole dimensions.

Secondary beam weld access hole dimensions



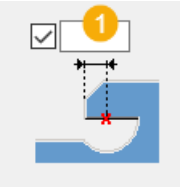
	Description
1	Vertical flange dimension.
2	Flange cut shape.
3	Weld access hole shape.
4	Radius of the weld access hole.

Beam end dimensions



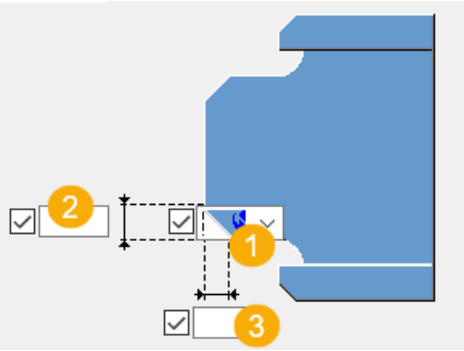
	Description
1	Length of the secondary beam web. This is the flange cut distance from the start of the web.
2	Vertical chamfer dimension.
3	Horizontal chamfer dimension.

Weld access hole offset



	Description
1	Weld access hole offset along the secondary beam axis.

Secondary beam web chamfers



	Description
1	Chamfer shape

	Description
2	Vertical chamfer dimension
3	Horizontal chamfer dimension

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

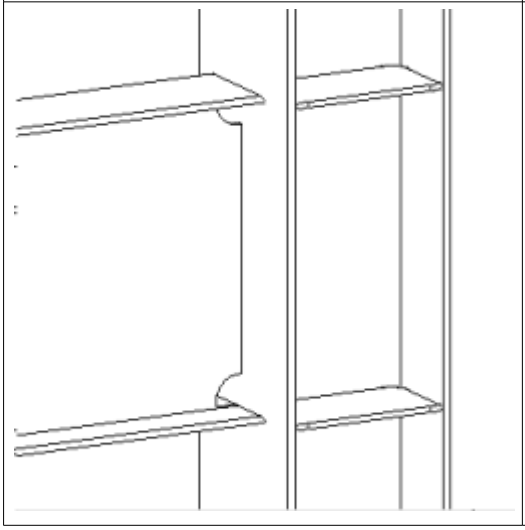
Welded column with stiffeners (128)

Creates a welded beam to column connection. By default, creates the necessary notches in the secondary beam flanges and web. The secondary beam can be level or sloped. Stiffener plates on the opposite side of the column web are optional.

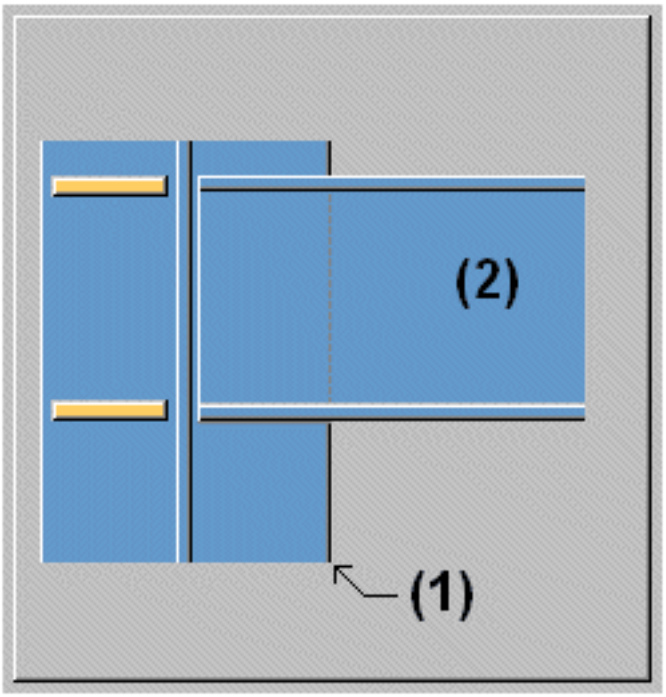
Objects created

- Stiffeners (optional)
- Weld backing bars (optional)
- Web doubler plate (optional)
- Bolts
- Welds
- Cuts

Use for

Situation	Description
	Welded to the column with weld prep and stiffener options.

Selection order



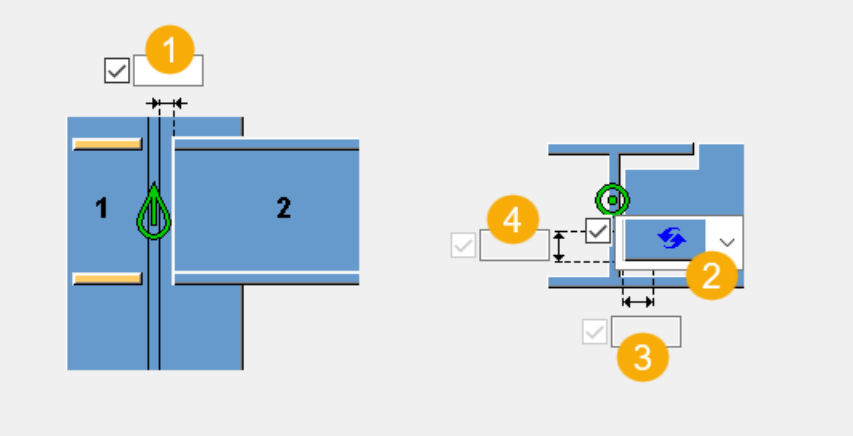
1. Select the main part (column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Picture tab

Use the **Picture** tab to control the gaps and shape of the secondary beam end cut.

Dimensions



	Description	Default
1	Cutting point of the secondary beam. The cutting point is defined as a gap between the main and the secondary part.	General / gap (0.0625”) 2 mm
2	Chamfer type. Options: No, Line, Cope, or Arc.	No chamfer
3	Width of the secondary flange chamfer.	Equal to the fillet radius of the main part profile.
4	Height of the secondary flange chamfer.	Equal to the fillet radius of the main part profile.

NOTE This information relates only to the U.S. imperial environment. `GENERAL` defaults can be found in the `joints.def` file in the system folder and can be modified as required.

Stiffeners tab

Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position, and type.

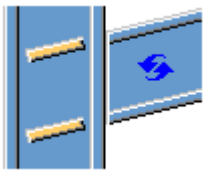
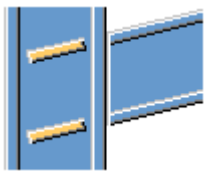
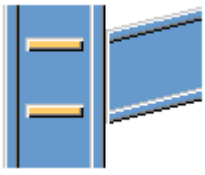
Stiffener plate dimensions

Part	Description
Top NS	Define the top near side stiffener thickness, width and height.
Top FS	Define the top far side stiffener thickness, width and height.

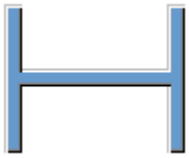
Part	Description
Bottom NS	Define the bottom near side stiffener thickness, width and height.
Bottom FS	Define the bottom far side stiffener thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Stiffener orientation

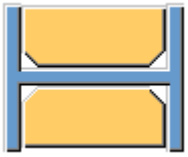
Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are parallel to the secondary part.
	Stiffeners are perpendicular to the main part.

Stiffener creation

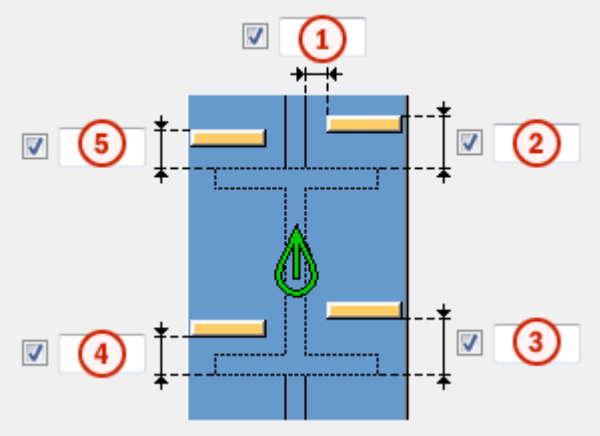
Option	Description
	Default Stiffeners are created. AutoDefaults can change this option.
	Automatic Stiffeners are created when necessary.
	No stiffeners are created.
	Stiffeners are created.

Stiffener shape

Option	Description
	Default Line chamfered stiffener plates AutoDefaults can change this option.
	Automatic Line chamfered stiffener plates
	Square stiffener plates Stiffener plates with a gap for the main part web rounding

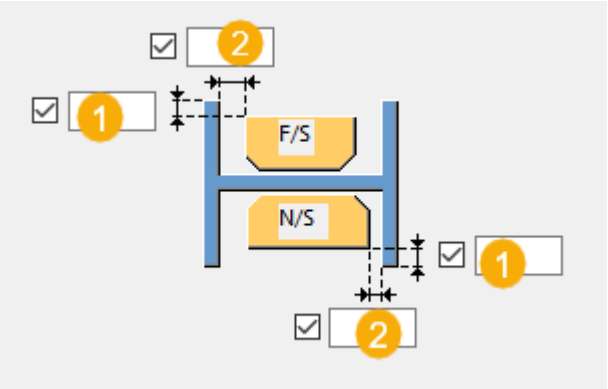
Option	Description
	Line chamfered stiffener plates

Stiffener positions



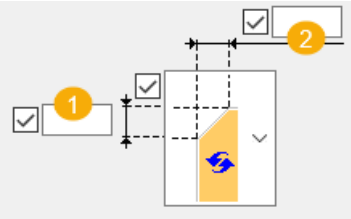
	Description
1	Size of the gap between the stiffener and the beam web edge.
2	Size of the gap between the top near side stiffener and the beam flange edge.
3	Size of the gap between the bottom near side stiffener and the beam flange edge.
4	Size of the gap between the bottom far side stiffener and the beam flange edge.
5	Size of the gap between the top far side stiffener and the beam flange edge.

Stiffener gap



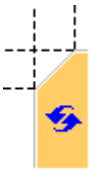
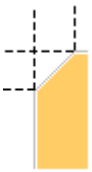
	Description
1	Distance from the edge of the flange to the edge of the stiffener.
2	Size of the gap between the flanges and the stiffener.

Chamfer dimensions



	Description	Default
1	Vertical dimension of the chamfer.	`10 mm`
2	Horizontal dimension of the chamfer.	`10 mm`

Chamfer type

Option	Description
	Default. Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer

Option	Description
	Concave arc chamfer

Beam cut tab

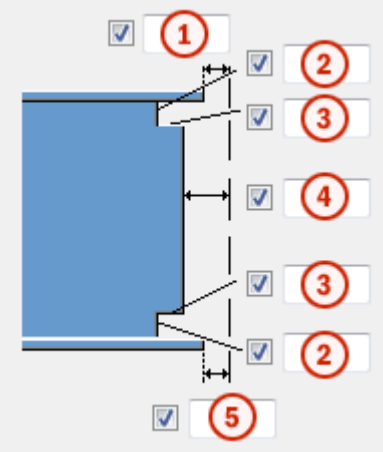
Use the **Beam cut** tab to control weld backing bars, weld access holes, beam end preparations, and flange cuts.

Weld backing bar

Option	Description
Weld backing bar	Weld backing bar thickness and width.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

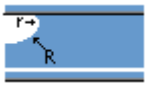
Weld access hole dimensions



	Description
1	Gap between the secondary part top flange and the main part.
2	Vertical dimensions for the top and the bottom weld access holes.
3	Horizontal dimensions for the top and the bottom weld access holes.
4	Gap between the secondary part web and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.
5	Gap between the secondary part bottom flange and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.

Weld access holes

Option	Description	Default
	Default Round weld access hole AutoDefaults can change this option.	
	Round weld access hole	
	Square weld access hole	
	Diagonal weld access hole	

Option	Description	Default
	Round weld access hole with a radius that you can define in r ☑	
	Extended cone-shaped weld access hole with a radius and dimensions that you can define in R ☑ and Top Prep x ☑ Bottom Prep x ☑	
	Cone-shaped weld access hole with radiuses that you can define in R ☑ and r ☑ Capital **R** defines the large radius (height). Small **r** defines the small radius.	R = 35 r = 10

Beam end preparation

Option	Description
	Default Top and bottom flange are prepared. AutoDefaults can change this option.
	Automatic Top and bottom flange are prepared.
	Beam end is not prepared.
	Top and bottom flange are prepared.

Flange cut

Option for top flange	Option for bottom flange	Description
		Default Flange is not cut. AutoDefaults can change this option.
		Flange is not cut.
		Flange is cut.

Weld backing bars

Option for top backing bar	Option for bottom backing bar	Description
		Default Backing bars are created inside the flanges. AutoDefaults can change this option.
		No backing bars are created.
		Backing bars are created inside the flanges.
		Backing bars are created outside the flanges.

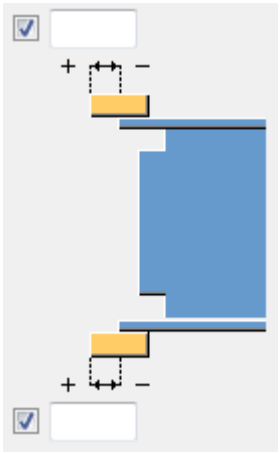
Weld backing bar length

Enter the length of the weld backing bar in the box below the options.

Option	Description
	Default Absolute length of the backing bar AutoDefaults can change this option.
	Absolute length of the backing bar

Option	Description
	Extension beyond the edge of the flange

Weld backing bar position

Option	Description
	Enter a positive or a negative value to move the front end of the backing bar relative to the end of the flange.

Assembly type

Define the location where the weld backing bar welds are made. When you select the **Workshop** option, Tekla Structures includes the backing bars in the assembly.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

Automatic notching options affect both the top and the bottom flange.

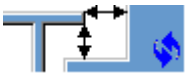
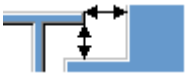
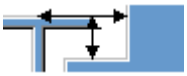
Notch shape

Automatic notching is switched on when you select a notch shape.

Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.

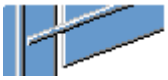
Option	Description
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

Enter the horizontal and vertical values for the cuts.

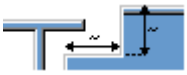
Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.

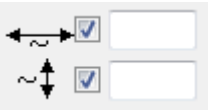
Option	Description
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.



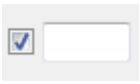
Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.



Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

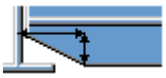
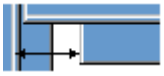
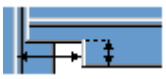
Side of flange notch

The side of flange notch defines on which side of the beam the notches are created.

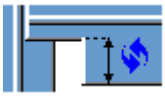
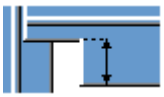
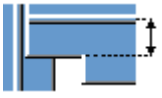
Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

Flange notch shape

The flange notch shape defines the notch shape in the beam flange.

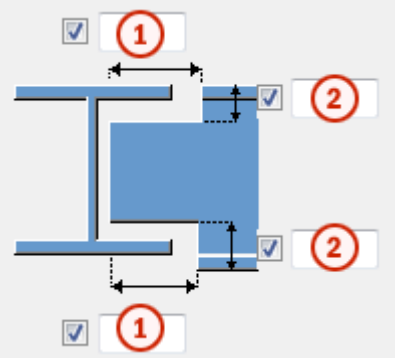
Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.

Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	`10 mm`
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest `5 mm`.

Doubler plate tab

Use the **Doubler plate** tab to create doubler plates to strengthen the web of the main part.

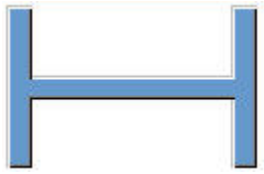
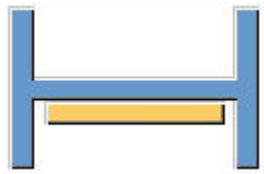
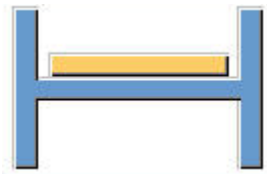
Web plate

Option	Description
Web plate	Web plate thickness and height.

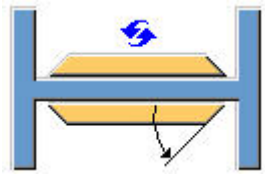
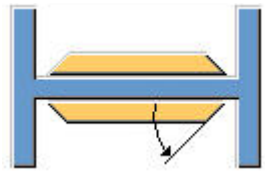
Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in

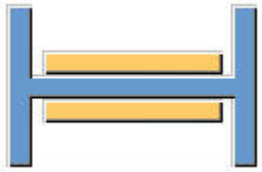
Option	Description	Default
		File menu --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Doubler plates

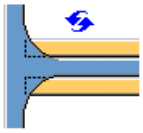
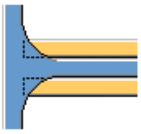
Option	Description
	Default Doubler plates are created on both sides. AutoDefaults can change this option.
	Doubler plates are not created.
	Doubler plate is created on the near side.
	Doubler plate is created on the far side.

Doubler plate edge shape

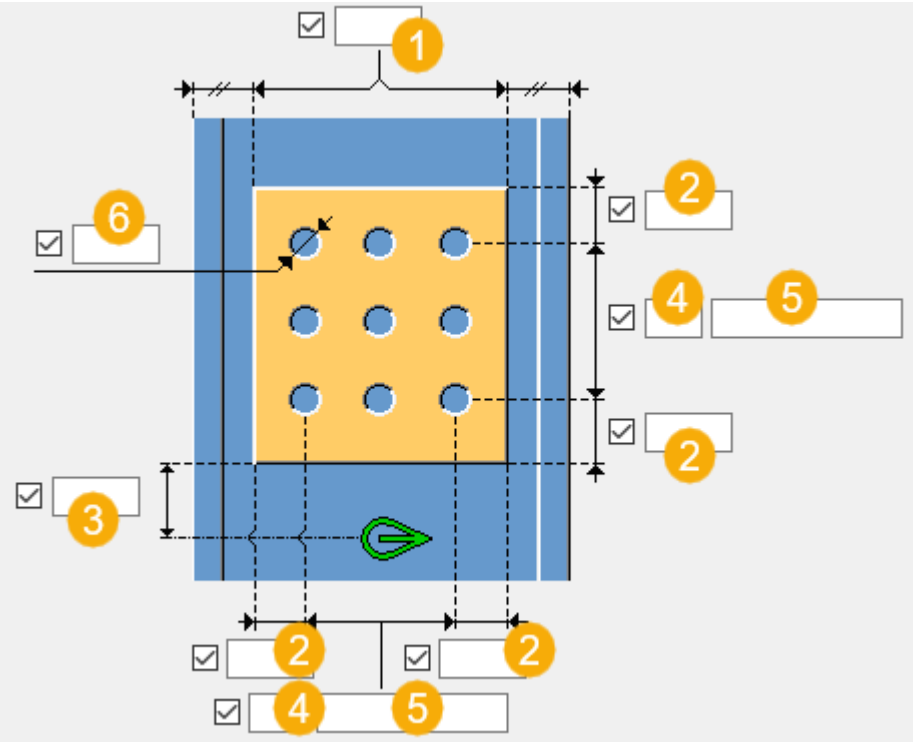
Option	Description
	Default Bevel doubler plates AutoDefaults can change this option.
	Bevel doubler plates Enter the angle in

Option	Description
	Square doubler plates

Doubler plate cuts

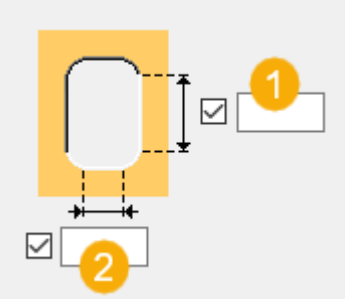
Option	Description
	Default Doubler plates are not cut. AutoDefaults can change this option.
	Doubler plates are not cut.
	Doubler plates are cut in the area that connects the main part web and flange.

General settings



	Description
1	Edge distance from the column flange.
2	Doubler plate edge distance. Edge distance is the distance from the center of a hole to the edge of the part.
3	Edge distance of the doubler plate in relation to the bottom of the secondary part.
4	Number of holes.
5	Hole spacing. Use a space to separate hole spacing values. Enter a value for each space between holes. For example, if there are 3 holes, enter 2 values.
6	Hole diameter.

Weld hole size



1	Slot length.
2	Slot width.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

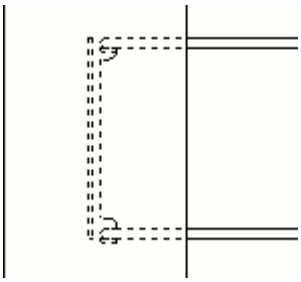
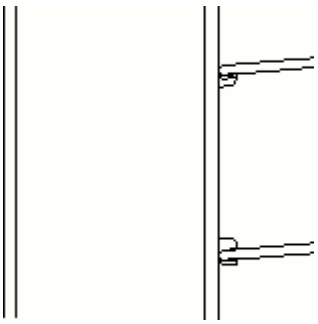
Beam prep (183)

Creates a welded beam to column connection. The end of the beam can be prepared. Web doubler plate in the column is optional. Welds are defined in a separate dialog box that you can access by clicking the **Welds** button.

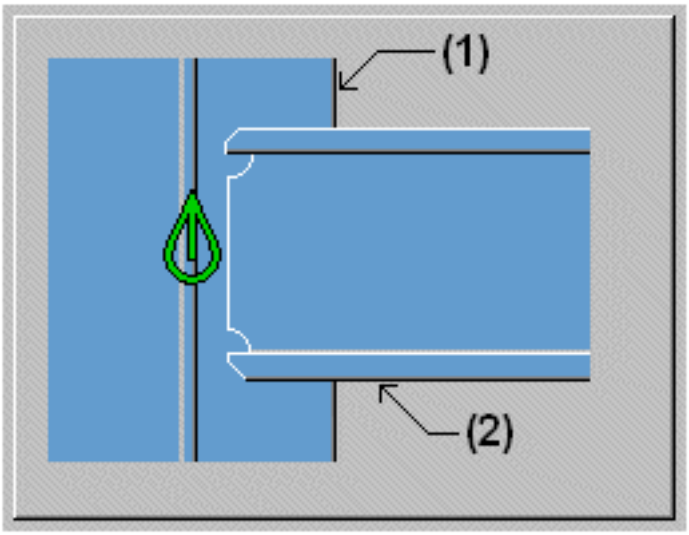
Objects created

- Web doubler plate (optional)
- Weld backing bars (optional)
- Welds
- Cuts

Use for

Situation	Description
	Beam to column web. Web doubler plate on the opposite side of column web.
	Sloping beam to column flange. Weld backing bars.

Selection order



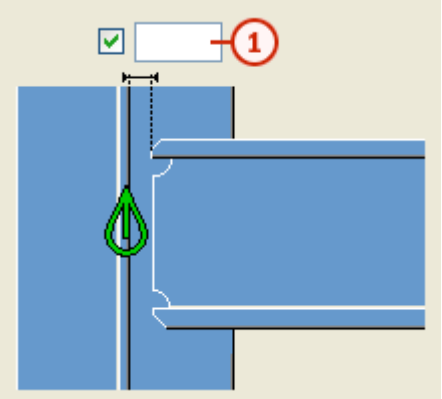
1. Select the main part (column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Picture tab

Use the **Picture** tab to set the gap between the main part and the secondary part, and select the beam end cut type.

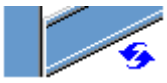
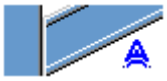
Dimensions



	Description
1	Weld gap

Beam end cut

Define how the beam end is cut. The secondary part is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Automatic If the secondary part slopes less than 10 degrees, a square cut is created. Otherwise, a bevel cut is made to the end of the secondary part.
	Square Creates a square cut to the end of the secondary part.
	Bevel Cuts the end of the secondary part parallel to the edge of the main part.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

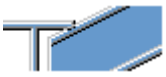
Automatic notching

Automatic notching options affect both the top and the bottom flange.

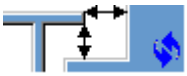
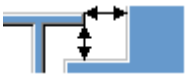
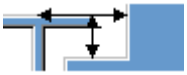
Notch shape

Automatic notching is switched on when you select a notch shape.

Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.

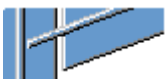
Option	Description
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

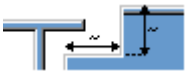
Enter the horizontal and vertical values for the cuts.

Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.

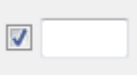
Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.



Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

Side of flange notch

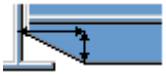
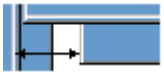
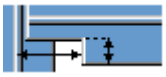
The side of flange notch defines on which side of the beam the notches are created.

Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

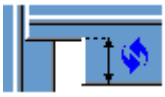
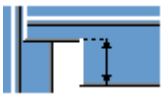
Flange notch shape

The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.

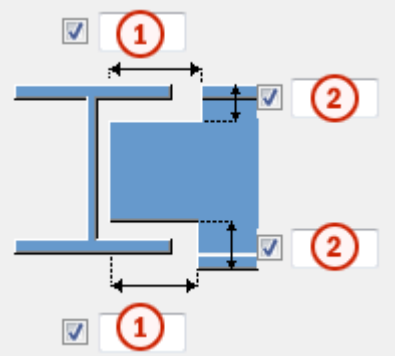
Option	Description
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.

Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	10 mm
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest 5 mm.

Beam cut tab

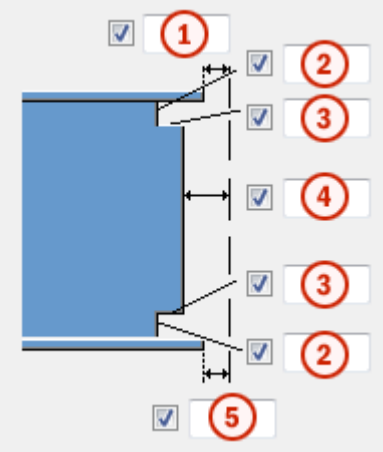
Use the **Beam cut** tab to control weld backing bars, weld access holes, beam end preparations, and flange cuts.

Weld backing bar

Option	Description
Weld backing bar	Weld backing bar thickness and width.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

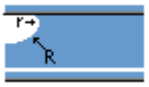
Weld access hole dimensions



	Description
1	Gap between the secondary part top flange and the main part.
2	Vertical dimensions for the top and the bottom weld access holes.
3	Horizontal dimensions for the top and the bottom weld access holes.
4	Gap between the secondary part web and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.
5	Gap between the secondary part bottom flange and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.

Weld access holes

Option	Description	Default
	Default Round weld access hole AutoDefaults can change this option.	
	Round weld access hole	
	Square weld access hole	
	Diagonal weld access hole	

Option	Description	Default
	Round weld access hole with a radius that you can define in r ☑	
	Extended cone-shaped weld access hole with a radius and dimensions that you can define in R ☑ and Top Prep x ☑ Bottom Prep x ☑	
	Cone-shaped weld access hole with radiuses that you can define in R ☑ and r ☑ Capital **R** defines the large radius (height). Small **r** defines the small radius.	R = 35 r = 10

Beam end preparation

Option	Description
	Default Top and bottom flange are prepared. AutoDefaults can change this option.
	Automatic Top and bottom flange are prepared.
	Beam end is not prepared.
	Top and bottom flange are prepared.

Flange cut

Option for top flange	Option for bottom flange	Description
		Default Flange is not cut. AutoDefaults can change this option.
		Flange is not cut.
		Flange is cut.

Weld backing bars

Option for top backing bar	Option for bottom backing bar	Description
		Default Backing bars are created inside the flanges. AutoDefaults can change this option.
		No backing bars are created.
		Backing bars are created inside the flanges.
		Backing bars are created outside the flanges.

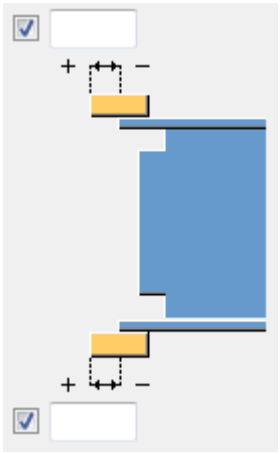
Weld backing bar length

Enter the length of the weld backing bar in the box below the options.

Option	Description
	Default Absolute length of the backing bar AutoDefaults can change this option.
	Absolute length of the backing bar

Option	Description
	Extension beyond the edge of the flange

Weld backing bar position

Option	Description
	Enter a positive or a negative value to move the front end of the backing bar relative to the end of the flange.

Assembly type

Define the location where the weld backing bar welds are made. When you select the **Workshop** option, Tekla Structures includes the backing bars in the assembly.

Doubler plate tab

Use the **Doubler plate** tab to create doubler plates to strengthen the web of the main part.

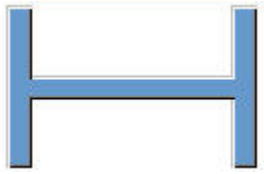
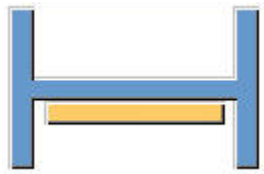
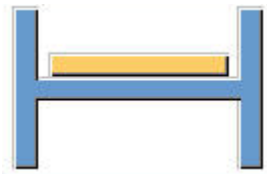
Web plate

Option	Description
Web plate	Web plate thickness and height.

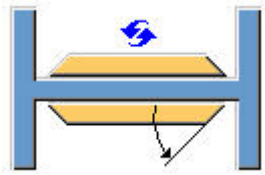
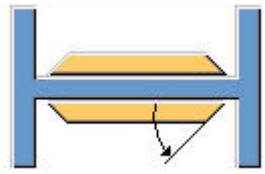
Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in

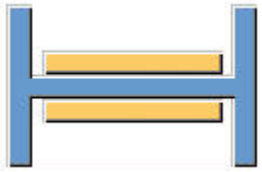
Option	Description	Default
		File menu --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Doubler plates

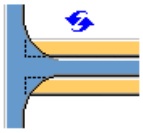
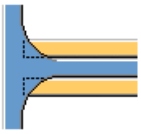
Option	Description
	Default Doubler plates are created on both sides. AutoDefaults can change this option.
	Doubler plates are not created.
	Doubler plate is created on the near side.
	Doubler plate is created on the far side.

Doubler plate edge shape

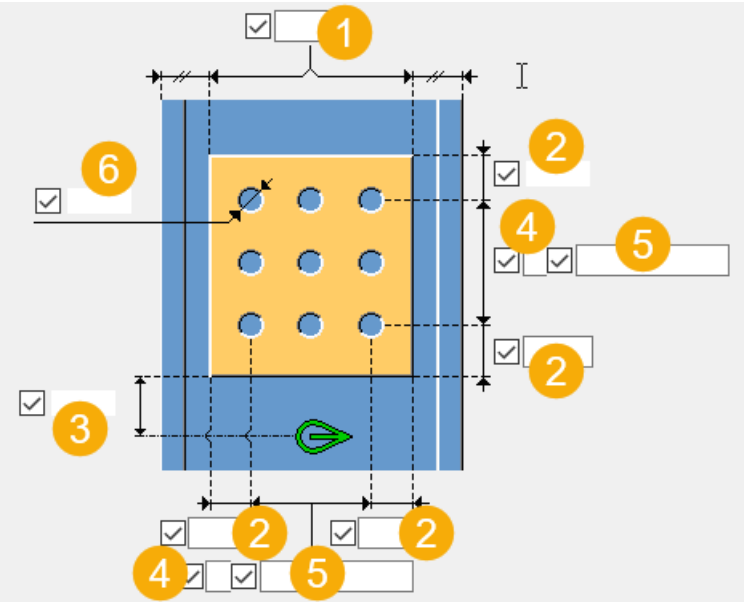
Option	Description
	Default Bevel doubler plates AutoDefaults can change this option.
	Bevel doubler plates Enter the angle in

Option	Description
	Square doubler plates

Doubler plate cuts

Option	Description
	Default Doubler plates are not cut. AutoDefaults can change this option.
	Doubler plates are not cut.
	Doubler plates are cut in the area that connects the main part web and flange.

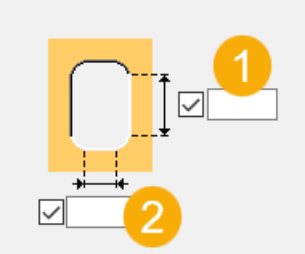
General settings



	Description
1	Edge distance from the column flange.

	Description
2	Doubler plate edge distance. Edge distance is the distance from the center of a hole to the edge of the part.
3	Edge distance of the doubler plate in relation to the bottom of the secondary part.
4	Number of holes.
5	Hole spacing. Use a space to separate hole spacing values. Enter a value for each space between holes. For example, if there are 3 holes, enter 2 values.
6	Hole diameter.

Weld hole size



Option	Description
1	Slot length.
2	Slot width.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

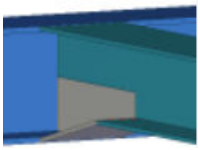
Offshore (194)

Offshore (194) creates a beam-to-beam welded connection, which also creates a haunch for different depth beams. The haunch is made of the same size profile as the smaller beam. You can control the length and depth of the cuts and the required welding.

Objects created

- Haunch plates
- Welds

Use for

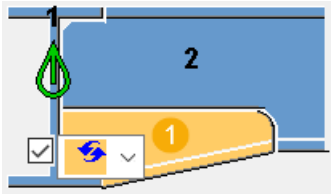
Situation	Description
	Welded offshore connection with haunch plates.

Selection order

1. Select the main part (beam).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

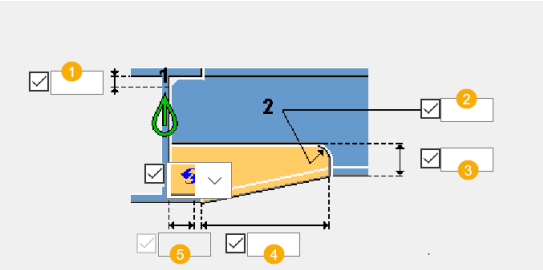


	Description
1	Haunch plate

Picture tab

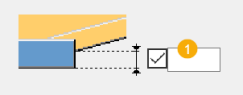
Use the **Picture** tab to define the connection dimensions and chamfers.

Dimensions



	Description
1	Chamfer size at the top flange.
2	Chamfer radius.
3	Depth of the secondary part cut.
4	Length of the secondary part cut.
5	Chamfer size at the bottom flange for line and convex chamfers.

Gap distance



	Description
1	Gap distance for the haunched beam flange.

Chamfer type

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer

Option	Description
	Convex chamfer

Parts tab

Use the **Parts** tab to define whether the haunch is created as a profile or as a plate.

Parts

Option	Description
Beam, **Web plate**, **Flange plates**	The haunch profile is the same as that of the supported secondary beam.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

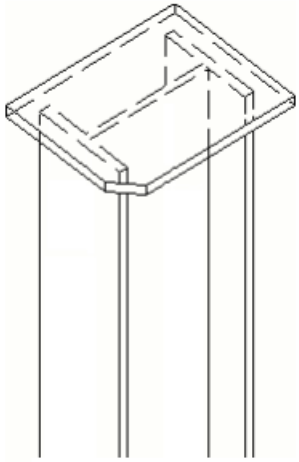
Cap plate detail

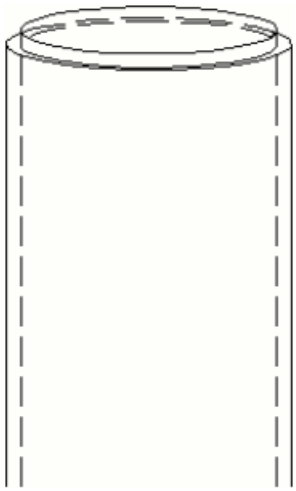
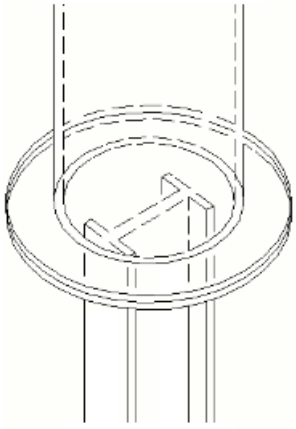
Cap plate detail creates a cap plate to a column or a beam, or to a polybeam.

Objects created

- Cap plate

Use for

Situation	Description
	Cap plate with one chamfered corner created on an I profile column.

Situation	Description
	Cap plate created on a pipe profile column.
	One cap plate created on a pipe profile column and another cap plate created on an I profile column.

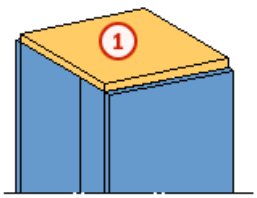
Selection order

1. Select the main part (beam, column, or polybeam).
2. Pick the position of the cap plate.

   You can pick a position on the column center line, side, or anywhere. The cap plate is always centered.

   The cap plate is created automatically when you pick the position.

Part identification key

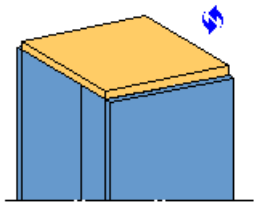
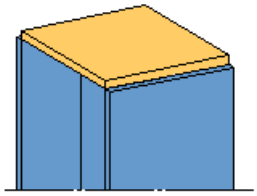
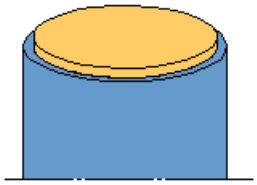


	Part
1	Cap plate

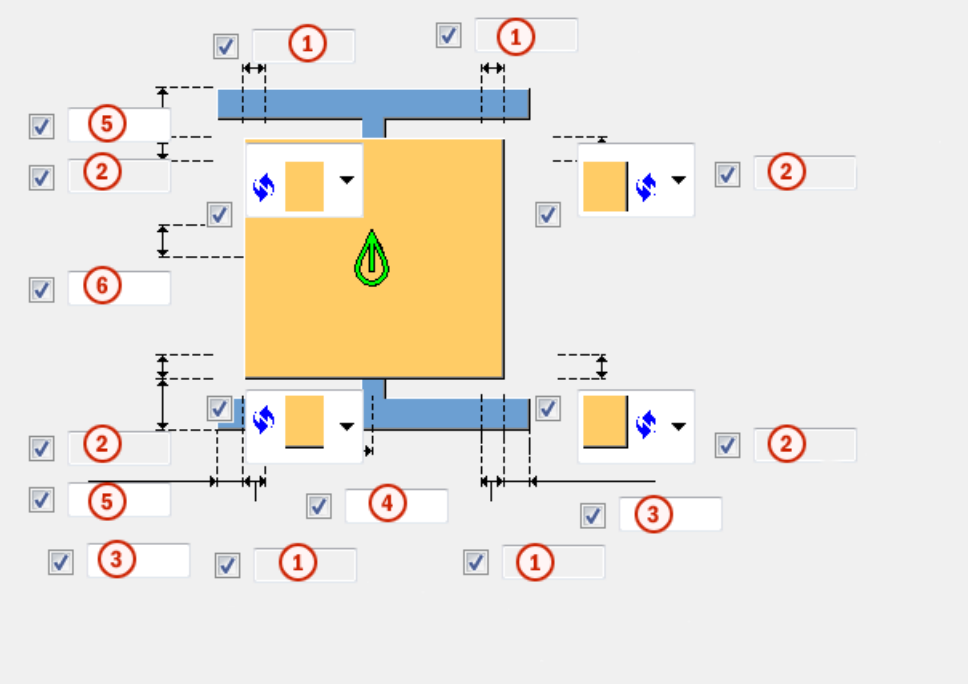
Picture tab

Use the **Picture** tab to control the shape and the dimensions the cap plate.

Cap plate shape

Option	Description
	Default Rectangular AutoDefaults can change this option.
	Rectangular
	Circular

Rectangular cap plate dimensions



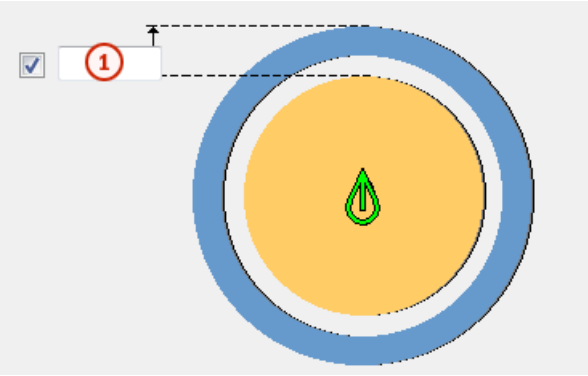
	Description	Default
1	Horizontal chamfer dimension. This option is not available if you have selected **No chamfer** in the chamfer options.	20 mm
2	Vertical chamfer dimension. This option is not available if you have selected **No chamfer** in the chamfer options.	20 mm
3	Horizontal edge distance of the cap plate. This option is used only when the plate width on the **Parts** tab is empty.	0 mm
4	Horizontal offset of the cap plate. This option is used only when the plate height on the **Parts** tab has a value.	0 mm
5	Vertical edge distance of the cap plate. This option is used only when the plate height on the **Parts** tab is empty.	10 mm
6	Vertical offset of the cap plate. This option is used only when the plate height on the **Parts** tab has a value.	0 mm

Rectangular cap plate chamfers

You can define the chamfers for each corner of the cap plate.

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Circular cap plate dimensions



	Description
1	Cap plate edge distance.

Parts tab

Use the **Parts** tab to control the properties of the cap plate.

Part

Part	Description	Default
Cap plate	Thickness, width, and height of the cap plate.	Thickness = 1.5*(profile web or plate thickness) Width = profile width or profile diameter or

Part	Description	Default
		height (option 'h'), first one that is not 0. Height = profile height or profile diameter or 200 mm, first one that is not 0.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

General tab

Click the link below to find out more:

General tab

Welds

Click the link below to find out more:

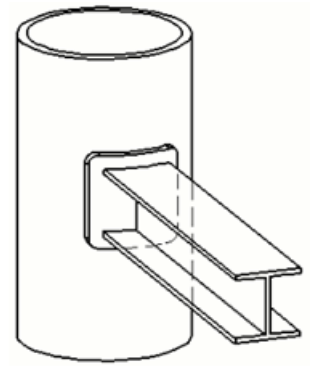
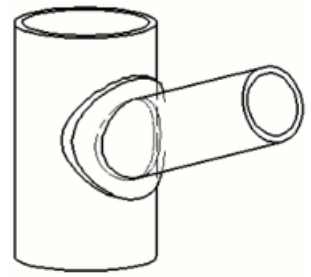
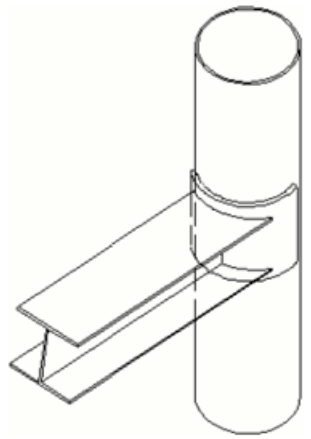
Pipe doubler plate

Pipe doubler plate creates a doubler plate between the main part and the secondary part. The main part has to be a round profile.

Objects created

- Doubler plate
- Welds

Use for

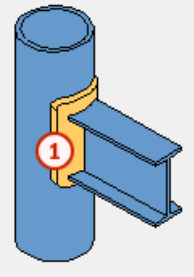
Situation	Description
	Rectangular doubler plate connecting a column and a beam.
	Circular doubler plate connecting a column and a beam. The secondary part is skewed.
	Rectangular doubler plate connecting a column and a beam. The secondary part is off-axis.

Selection order

1. Select the main part (column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

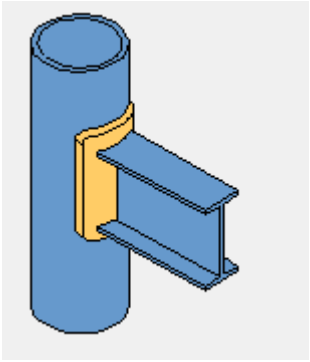
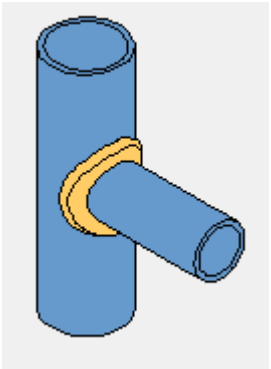


	Part
1	Doubler plate

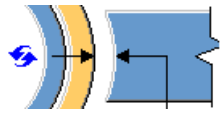
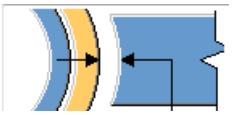
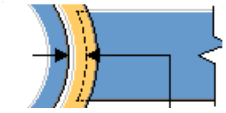
Picture tab

Use the **Picture** tab to control the type, position, and dimensions of the doubler plate.

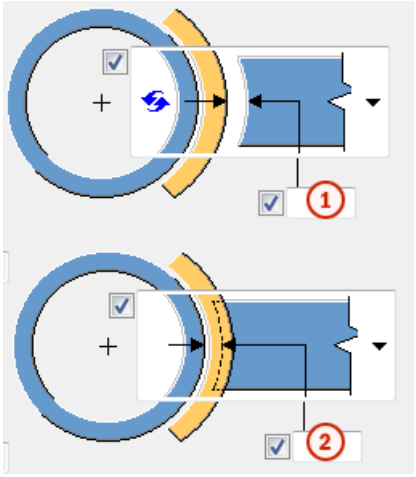
Doubler plate type

Option	Description
 	Select the doubler plate type: • **Rectangular** • **Circular**

Clearance type

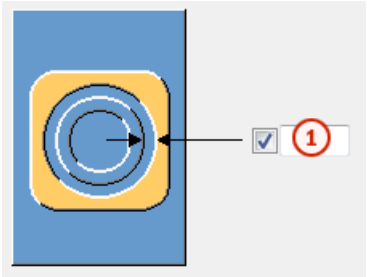
Option	Description
	Default Clearance between the doubler plate and the secondary part. AutoDefaults can change this option.
	Clearance between the doubler plate and the secondary part.
	Clearance between the main part and the secondary part.

Clearance dimension



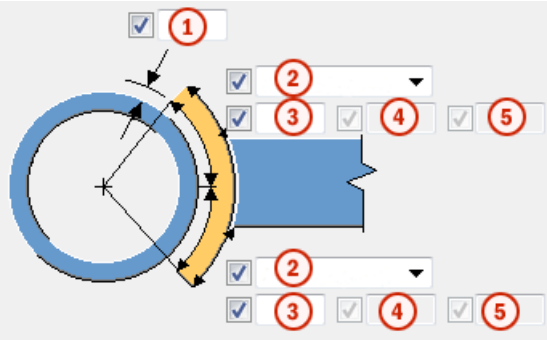
	Description
1	Clearance between the doubler plate and the secondary part.
2	Clearance between the main part and the secondary part.

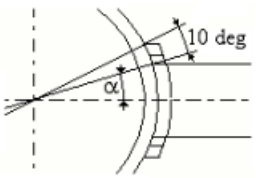
Cut clearance



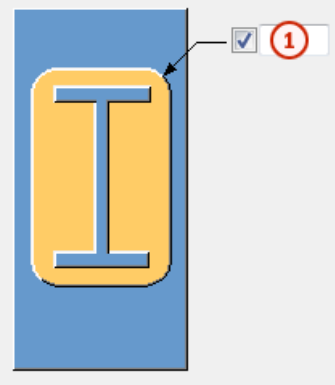
	Description
1	Cut clearance between the doubler plate and the secondary part.

Rectangular doubler plate dimensions



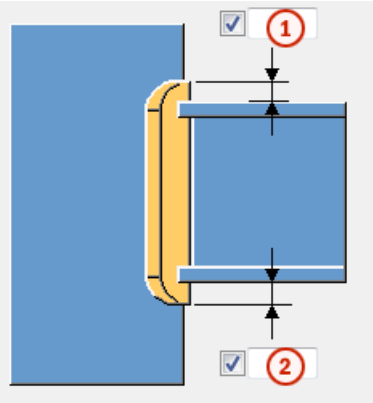
	Description	Default
1	Offset between the column and the doubler plate.	`0 mm`
2	Measuring type for the doubler plate: • Angle (degrees) Define the angle of the doubler plate in degrees. • Dimension Define the arc length of the doubler plate. • Edge Define the arc length of the doubler plate edge.	Angle
3	Angle of the doubler plate. The angle can be extended up to 360 degrees to go around the main part.	Angle +10 degrees 
4	Dimension value of the doubler plate. The dimension can be extended up to 360 degrees to go around the main part.	`length equivalent to the formula angle +10 degrees`
5	Edge value of the doubler plate. The edge value can be extended up to 360 degrees to go around the main part.	`length equivalent to the formula angle +10 degrees`

Corner rounding dimension for rectangular plate



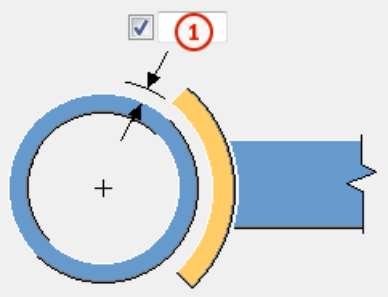
	Description
1	Corner rounding dimension for a rectangular doubler plate. Corners are not created with value `0`.

Doubler plate extension for rectangular plate



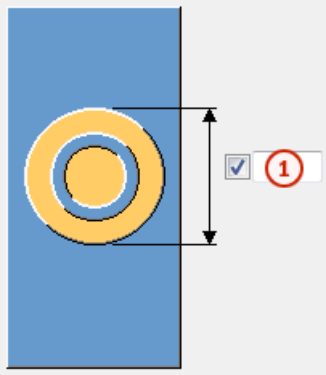
	Description
1	Vertical dimension from the secondary part top edge to doubler plate top edge.
2	Vertical dimension from the secondary part bottom edge to the doubler plate bottom edge.

Circular doubler plate dimensions



	Description
1	Offset between the column and the doubler plate.

Circular doubler plate diameter



	Description
1	Diameter of the circular doubler plate.

Parts tab

Use the **Parts** tab to control the position of the doubler plate.

Part

Part	Description
Doubler plate	Thickness of the doubler plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
	assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

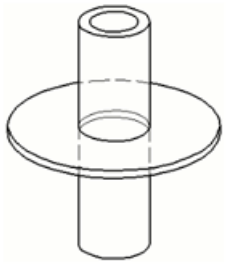
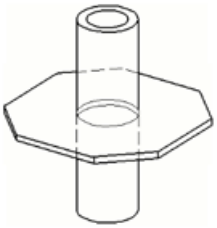
Ring plate

Ring plate creates a ring plate. The ring plate can be circular, elliptical, or angular. **Ring plate** creates welds between the column and the ring plate.

Objects created

- Ring plate
- Welds

Use for

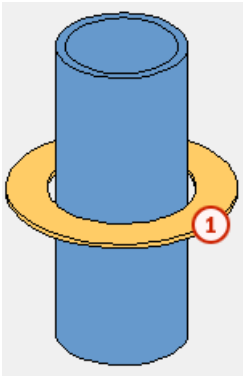
Situation	Description
	Circular ring plate on a tube column.
	Angular ring plate on a tube column.

Selection order

1. Select the main part (column).
2. Pick a position for the connection.

   The connection is created automatically when the position is picked.

Part identification key



	Part
1	Ring plate

Picture tab

Use the **Picture** tab to control the shape and the dimensions of the ring plate in the **Ring plate** connection.

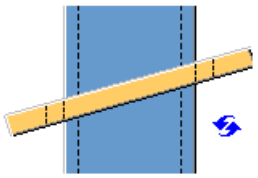
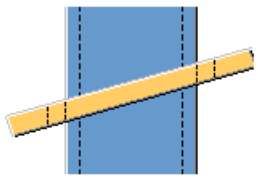
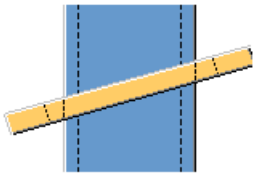
Ring plate type

Option	Description	Default
Ring plate	Select the ring plate type: • **Default** • **Circular** • **Elliptical** • **Angular**	Circular

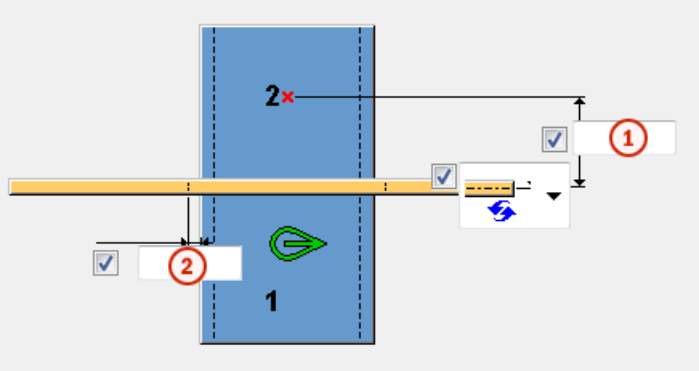
Ring plate measuring type

Option	Description	Default
Ring plate measuring type	Select how the ring plate dimension is measured: • **Default** • **Plate dimension** • **Plate absolute dimension**	Plate dimension

Cut direction

Option	Description
	Default The cut is created along the main part. AutoDefaults can change this option.
	The cut is created along the main part.
	The cut is created perpendicular on the plate.

Ring plate offset dimensions



	Description
1	Ring plate offset from the picked point.
2	Ring plate clearance from the outer edge of the main part.

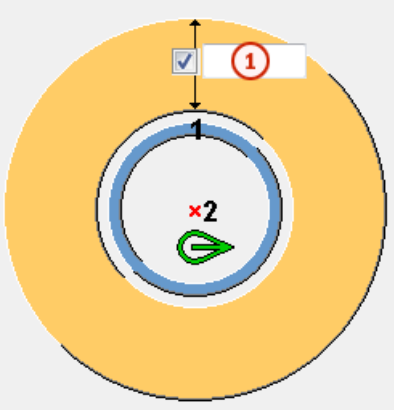
Ring plate offset measuring

Option	Description
	Default The offset dimension is measured from the center line of the ring plate. AutoDefaults can change this option.
	The offset dimension is measured from the center line of the ring plate.
	The offset dimension is measured from the bottom edge of the ring plate.
	The dimension is measured from the top edge of the ring plate.

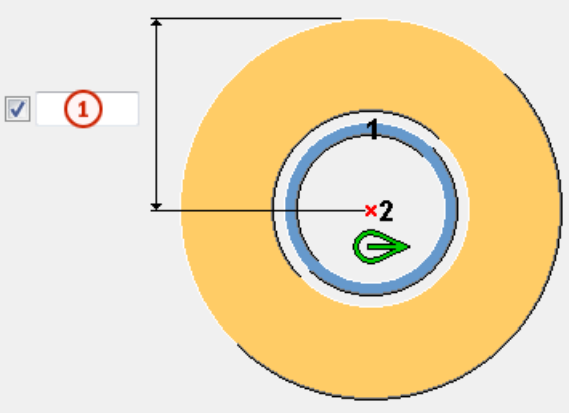
Create assembly

Option	Description	Default
Create assembly	Define how the parts are included in an assembly: • Default • **Yes** - All parts are in one single assembly. • **No** - Parts are not included in one single assembly.	**Yes**

Circular ring plate dimensions

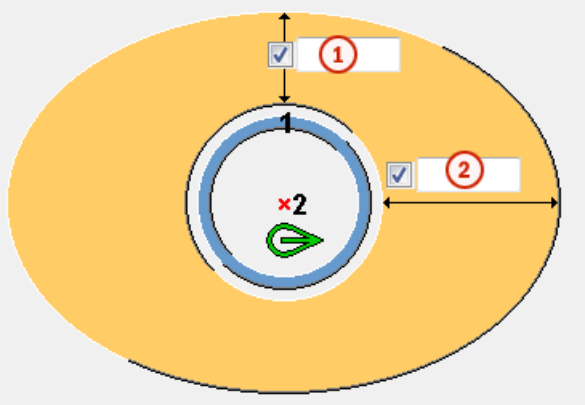


	Description	Default
1	**Plate dimension** Define the minimum width of the ring plate.	`200mm`

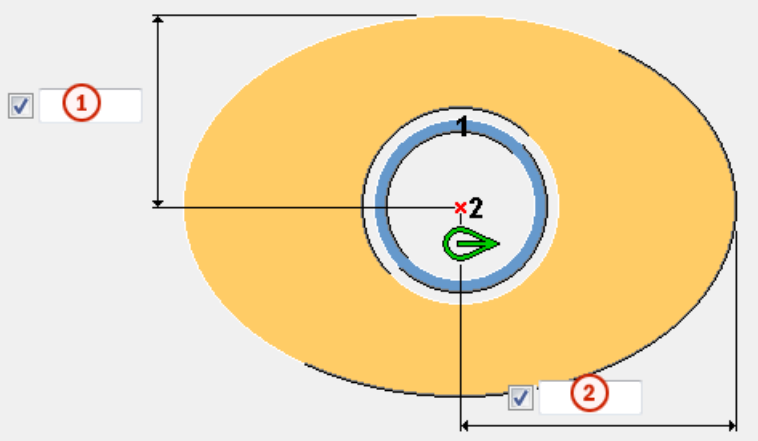


	Description	Default
1	**Plate absolute dimension** Define the absolute width of the ring plate from the center point.	`Minimum width + clearance + part height/2`

Elliptical ring plate dimensions



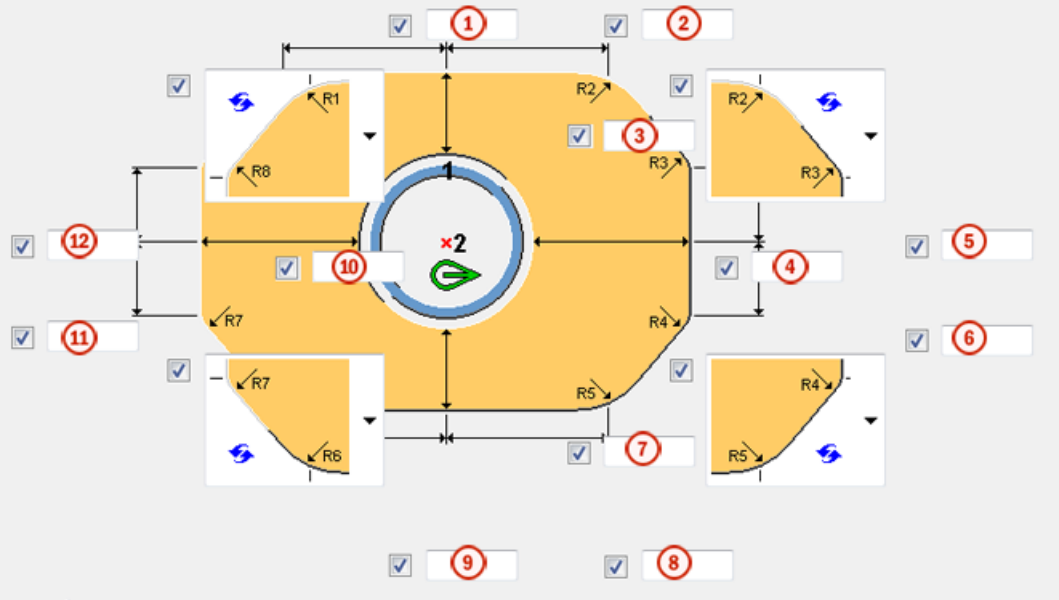
	Description	Default
1	Ring plate vertical width.	`200mm`
2	Ring plate horizontal width.	`200mm`



	Description	Default
1	Absolute vertical width of the ring plate from the center point.	`Vertical width + clearance + part height/2`
2	Absolute horizontal width of the ring plate from the center point.	`Horizontal width + clearance + part height/2`

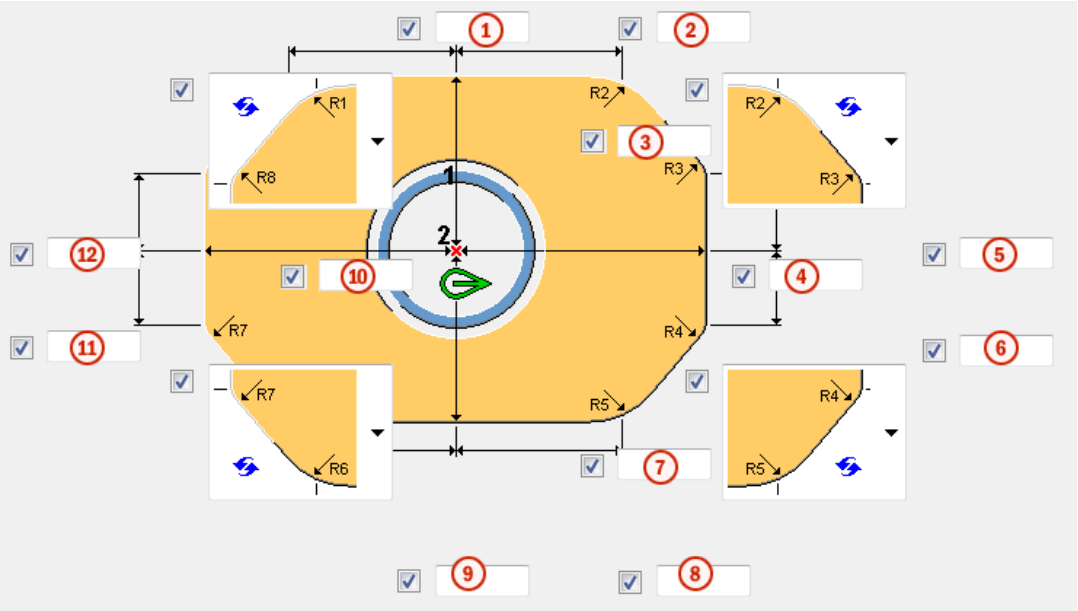
Angular ring plate dimensions

You can define the radius for each corner.



	Description	Default
1	Horizontal distance between corner radius R1 and the picked point.	150mm
2	Horizontal distance between corner radius R2 and the picked point.	150mm
3	Vertical distance between the plate top outer edge and the plate inner edge.	200mm
4	Horizontal distance between the plate right side edge and the plate inner center.	200mm
5	Vertical distance between corner radius R3 and the picked point.	150mm
6	Horizontal distance between corner radius R4 and the picked point.	150mm
7	Vertical distance between the plate bottom outer edge and the plate inner center.	200mm
8	Horizontal distance between corner radius R5 and the picked point.	150mm
9	Horizontal distance between corner radius R6 and the picked point.	150mm

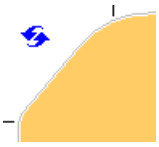
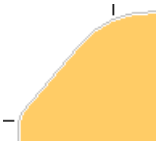
	Description	Default
10	Horizontal distance between the plate left side edge and the plate inner edge.	`200mm`
11	Vertical distance between corner radius R7 and the picked point.	`150mm`
12	Vertical distance between corner radius R8 and the picked point.	`150mm`



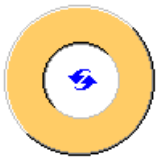
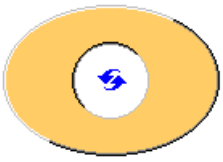
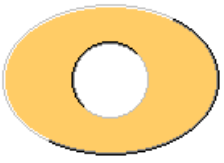
	Description	Default
1	Horizontal distance between corner radius R1 and the picked point.	`150mm`
2	Horizontal distance between corner radius R2 and the picked point.	`150mm`
3	Absolute vertical distance between the plate top outer edge and the center point.	`Vertical width + clearance + part height/2`
4	Absolute horizontal distance between the plate right side edge and the center point.	`Horizontal width + clearance + part height/2`
5	Vertical distance between corner radius R3 and the picked point.	`150mm`
6	Horizontal distance between corner radius R4 and the picked point.	`150mm`
7	Absolute vertical distance between the plate bottom outer edge and the center point.	`Vertical width + clearance + part height/2`

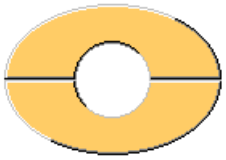
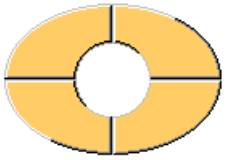
	Description	Default
8	Horizontal distance between corner radius R5 and the picked point.	`150mm`
9	Horizontal distance between corner radius R6 and the picked point.	`150mm`
10	Absolute horizontal distance between the plate left side edge and the center point.	`Horizontal width + clearance + part height/2`
11	Vertical distance between corner radius R7 and the picked point.	`150mm`
12	Vertical distance between corner radius R8 and the picked point.	`150mm`

Corner shape

Option	Description
	Default Two radius dimensions AutoDefaults can change this option.
	Two radius dimensions
	One radius dimension

Ring plate

Option (circular)	Option (elliptical)	Option (angular)	Description
			Default One ring plate AutoDefaults can change this option.
			One ring plate

Option (circular)	Option (elliptical)	Option (angular)	Description
			Two half ring plates
			Four quarter ring plates

Parts tab

Use the **Parts** tab to control the properties of the ring plate.

Part

Part	Description	Default
Ring plate	Thickness of the ring plate.	10mm

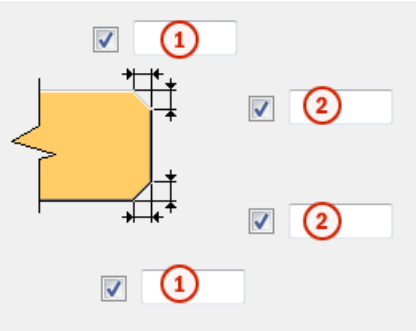
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Chamfers tab

Use the **Chamfers** tab to control the chamfer properties of the ring plate.

Chamfer dimensions

For circular and elliptical ring plates, chamfering is created uniformly around the outer perimeter of the ring plate. For angular ring plates, chamfers are created in each corner.



	Description
1	Horizontal dimension of the chamfer.
2	Vertical dimension of the chamfer.

General tab

Click the link below to find out more:

General tab

Welds

Click the link below to find out more:

5.7 Seated connections

This section introduces components that can be used in seated connections.

- Seating (39) (page 1816)
- U.S. Seat connection 3 (74) (page 1829)
- Angle profile box (170) (page 1844)

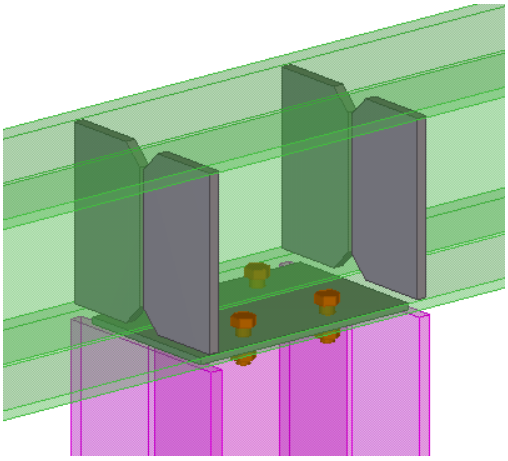
Seating (39)

Seating (39) connects a column head to one or two beams using an end plate. The end plate is welded to the column head and bolted to the lower flange of the secondary part.

Objects created

- End plate
- Stiffeners
- Bolts
- Washer plates (optional)

Use for

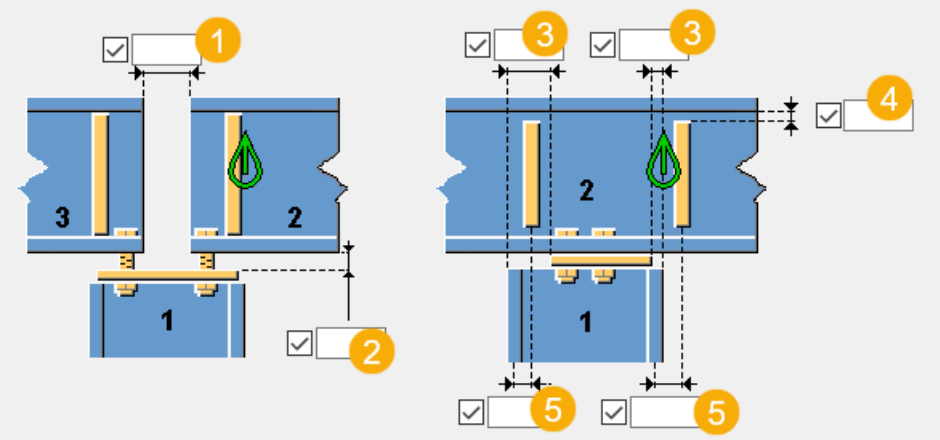
Situation	Description
	Seating connection with end plate and stiffeners.

Selection order

1. Select the main part (column).
2. Select the first secondary part (beam).
3. Select the second secondary part (beam).
4. Click the middle mouse button to create the connection.

Picture tab

Use the **Picture** tab to control the end plate and stiffener size and position.



	Description
1	Distance between the first and the second secondary part.
2	Distance between the end plate the secondary parts.
3	End plate distance from the main part edge. Positive values move end plate edges closer to the column axis and thus decrease the plate size. Negative values increase the plate size. The default value is `10 mm`.
4	A gap between stiffener plates and secondary beam flange. The default value is `0 mm`.
5	Position of the stiffeners. By default, stiffeners are placed to the same plane as column flanges. Positive offset values move stiffeners to the right and negative to the left.

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Square AutoDefaults can change this option.
	Square Cuts the end of the secondary beam square.
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.

Parts tab

Use the **Parts** tab to control the end plate, stiffener and washer plate properties.

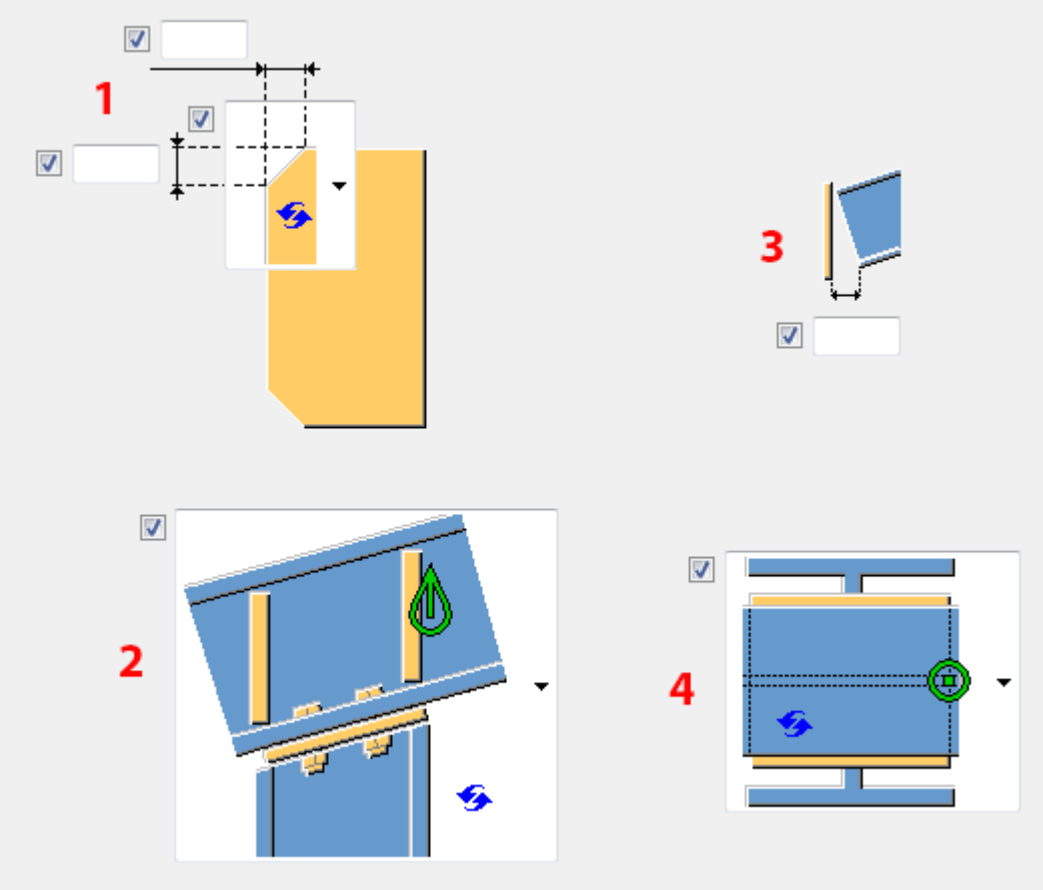
Define the end plate, stiffener and washer plate thickness, width and height.

Option	Description
End plate	By default, width is defined by the bolt group horizontal edge distances, and height is defined by the plate edge distances from the left and right edge of the column. The default value for end plate thickness is `0.5*the screw diameter`.
Stiffeners	By default, height is equal to the distance between secondary beam flanges. If width is not given, stiffener width is defined based on the flange width. The default value for the stiffener thickness is `1.5*secondary beam web thickness` rounded upwards to: 8, 10, 12, 16, 20, 25, 30, 35, 40, 45, and so on.
Washer plates	Washer plates are small rectangular plates used as washers between the bolt head and the secondary beam flange. If no thickness is defined, plates are not created.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Parameters tab

Use the **Parameters** tab to control the chamfer dimensions, and type and the end plate and stiffener orientation.



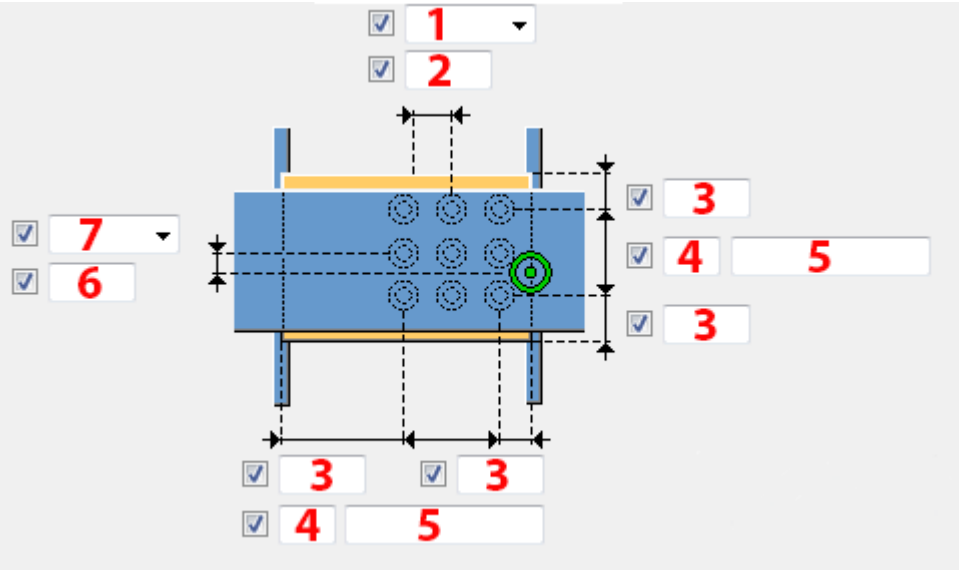
	Description
1	Chamfer type and dimensions. If you select the No chamfer option, a clash can occur between the stiffener and the I-profile rounding. Additionally, you can define the chamfer dimensions vertically and horizontally. If you select an arc chamfer, the horizontal dimension is the radius, and the vertical dimension has no effect.
2	Select whether stiffeners are perpendicular or parallel to the secondary beam flange.
3	Gap size to end plate. Define the limit value for the gap between the end plate and the secondary or main part. Use this gap when the beam is slightly curved

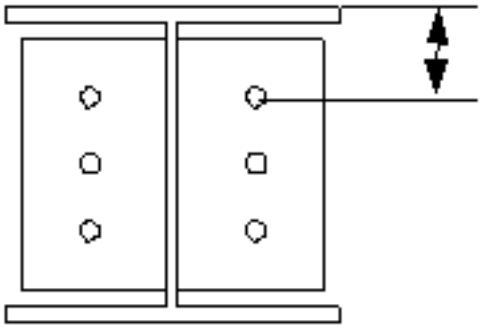
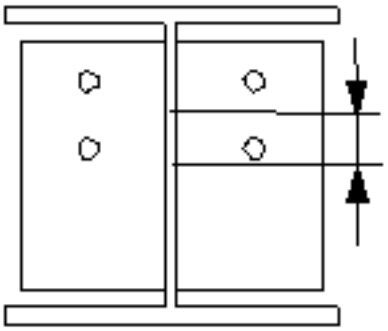
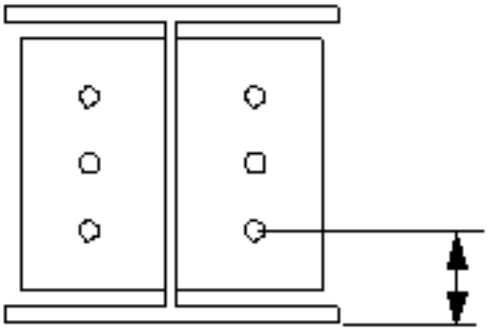
	Description
	or sloped to decide if the end angle is so small that the beam end can be straight. If the actual gap is smaller than this value, the beam end is left straight. If the actual gap is larger than this value, the beam end is fitted to the end plate.
4	Select the end plate orientation.

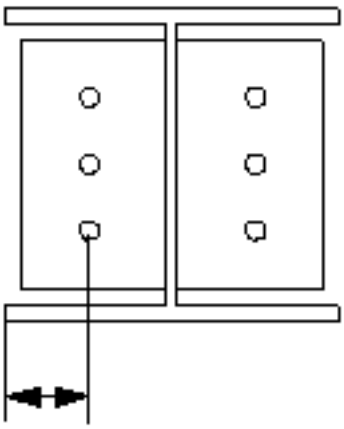
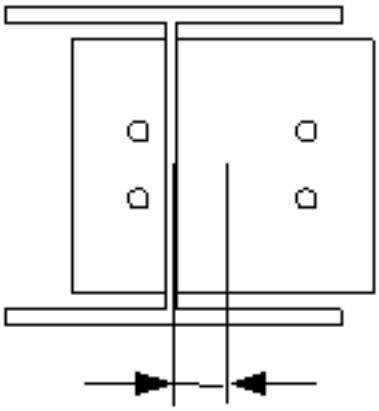
Bolts tab

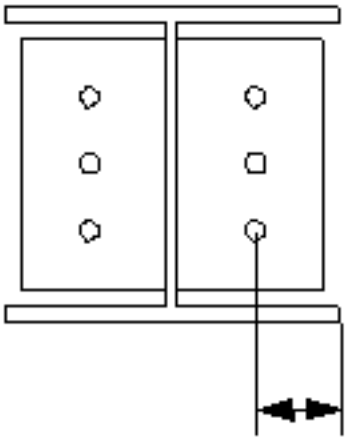
Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions



1	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
2	Dimension for vertical bolt group position.

3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Dimension for horizontal bolt group position.
7	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt.  • **Middle**: From the center line of the secondary part to the center line of the bolts. 

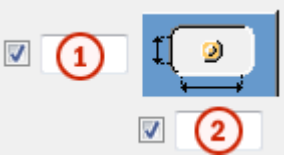
	• **Right**: From the right edge of the secondary part to the rightmost bolt. 

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

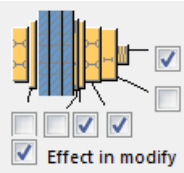


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

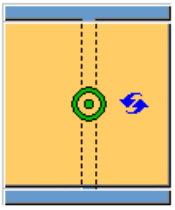
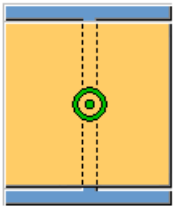
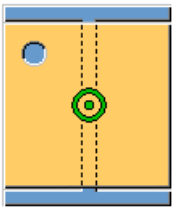
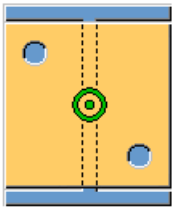
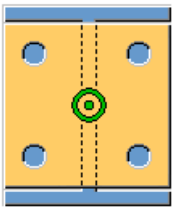
Holes tab

Use the **Holes** tab to control the galvanizing holes in the end plate.

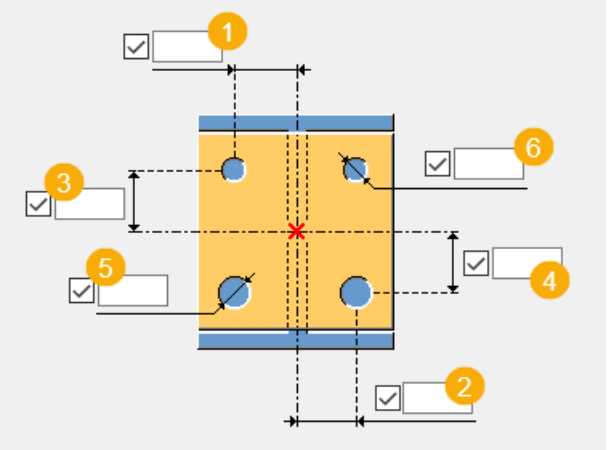
Option	Description
Bolt standard	Select the bolt standard.
Bolt type	Select the bolt type to define the location where the bolts should be attached.
Read data from	You can select to use the `sinkholes.dat` definition file to specify the default values for horizontal and vertical offsets, and the diameters for upper and lower holes. The file is searched in the following order: Environment common system steel folder (`..\Environments\common\system\Steel`), model folder, `XS_FIRM`, `XS_PROJECT` and `XS_SYSTEM` folder. You can also select to define the holes in the component dialog box.

Number of holes

The center of a hole group is the middle point of the beam and the middle point of the haunch, if the haunch exists. The hole groups are composed of 0, 1, 2 or 4 holes.

Option	Description
	Default No holes AutoDefaults can change this option.
	No holes
	1 hole
	2 holes
	4 holes

Hole positions



	Description
1	Horizontal distance between the secondary beam center and the upper hole.
2	Horizontal distance between the secondary beam center and the lower hole.
3	Vertical distance between the secondary beam center and the upper hole.
4	Vertical distance between the secondary beam center and the lower hole.
5	Diameter of the lower hole.
6	Diameter of the upper hole.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

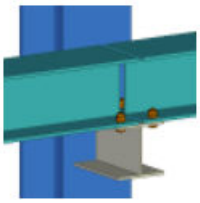
U.S. Seat connection 3 (74)

U.S. Seat connection 3 (74) connects two beams to a column when the beams are offset from the column center line. The seat is always placed perpendicular to the beams. You can use the connection with rotated and skewed beams and columns. The seat can be bolted or welded to the beam, but it is always welded to the column.

Objects created

- Seat profile
- Stiffeners
- Bolts
- Welds

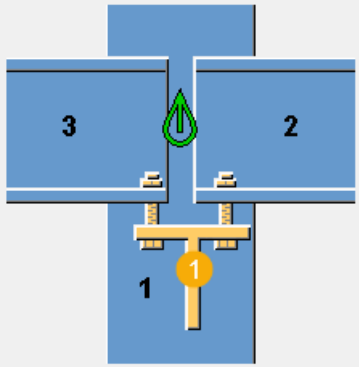
Use for

Situation	Description
	Seat profile welded to a column and bolted to two beams.

Selection order

1. Select the main part (column).
2. Select the first secondary part (beam).
3. Select the second secondary part (beam).
4. Click the middle mouse button to create the connection.

Part identification key

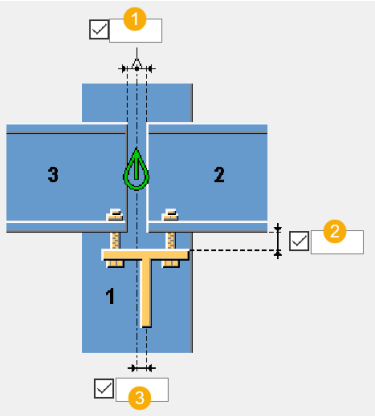


	Description
1	Seat profile

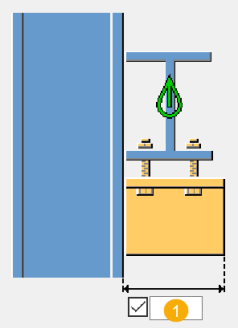
Picture tab

Use the **Picture** tab to define the offset dimensions of the connection.

Offset dimensions



	Description
1	Offset of the beams from the starting point of the connection
2	Offset of the seat from the beams
3	Offset of the seat from the center of the connection



	Description
1	Length of the seat

Parts tab

Use the **Parts** tab to define the part properties.

Parts

Option	Description	Default
Stiffeners	Thickness, width and height of the stiffeners. The stiffener height is in the same direction as the column.	The default values for the height and width are based on the seat profile dimensions. The default stiffener thickness is ¼" or 6 mm depending on whether metric or imperial units are used in the model. Stiffeners can only be placed if an angle profile is used for the seat.
Seat profile	Select the profile from the profile catalog.	WT6X15 tee

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

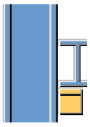
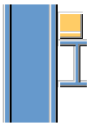
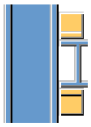
Option	Description	Default
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

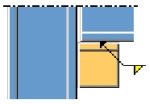
Use the **Parameters** tab to define seat position and orientation, and stiffener position, shape, and chamfering.

Option	Description
Seat notching and fitting	Select how the seat is notched and fitted to the column.
Cut tolerance of sec	Define a cut tolerance of the secondary part.

Seat position

Option	Description
	Default Seat is placed at the bottom of the beam. AutoDefaults can change this option.
	Seat is placed at the bottom of the beam.
	Seat is placed at the top of the beam.
	Seat is placed at both the top and bottom of the beam.

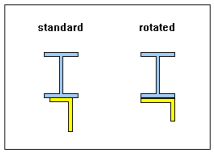
Seat to beam

Option	Description
	Default Seat is bolted to the beam. AutoDefaults can change this option.
	Seat is bolted to the beam.
	Seat is welded to the beam.

Seat orientation

Option	Description
	Default Standard seat, not rotated. AutoDefaults can change this option.
	Standard seat, not rotated.
	Seat is rotated.

If the seat is an angle with uneven legs, the standard option places the shortest leg of the angle against the beam. Selecting rotated reverses this.



Beam fitting

Option	Description
	Default Beam is not fitted to the column. AutoDefaults can change this option.
	Beam is not fitted to the column.

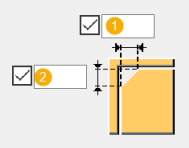
Option	Description
	Beam is fitted to the column.

Stiffener shape

Stiffeners are only created if an angle profile is used for the seat.

Option	Description
	Default Rectangular stiffener AutoDefaults can change this option.
	Rectangular stiffener
	Triangular stiffener

Stiffener chamfer dimensions



	Description
1	Horizontal chamfer dimension
2	Vertical chamfer dimension

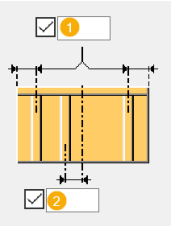
Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	Line chamfer
	Convex chamfer
	Concave chamfer

Stiffener positions

Option	Description
	Default Stiffeners are not placed on the seat. AutoDefaults can change this option.
	Stiffeners are not placed on the seat.
	Stiffener is placed on the right side.
	Stiffener is placed in the middle.
	Stiffener is placed on the left side.

Stiffener offsets

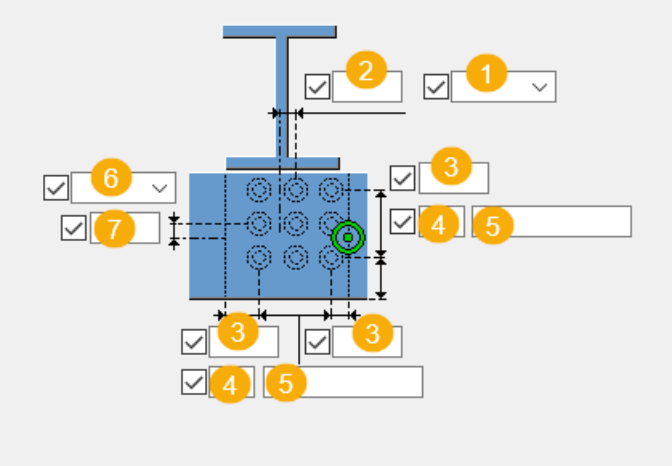


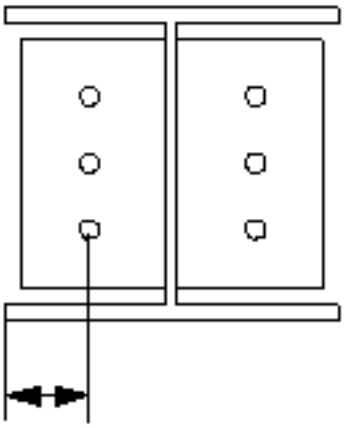
	Description
1	Offset of the end stiffeners from the seat ends.
2	Offset of the center stiffener from the center line of the seat.

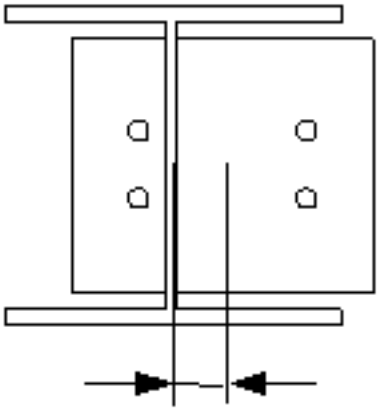
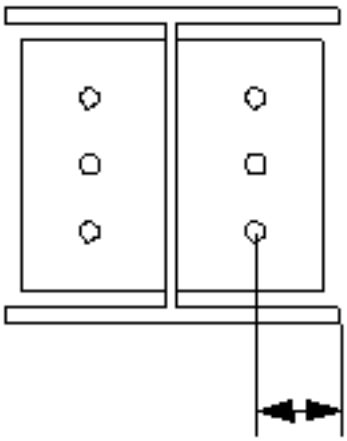
Bolts tab

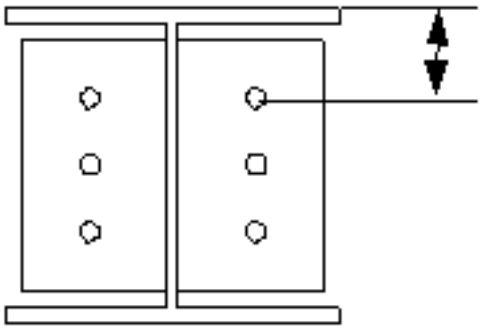
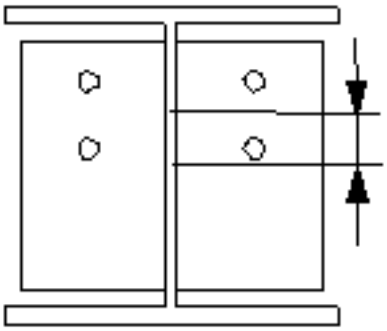
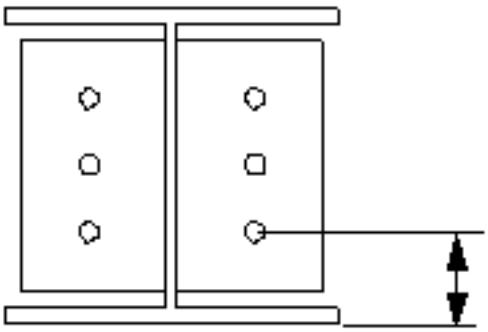
Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt. 

	Description
	• **Middle**: From the center line of the secondary part to the center line of the bolts.  • **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

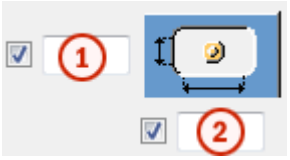
	Description
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Dimension for vertical bolt group position.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



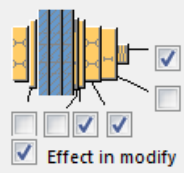
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	

Option	Description	Default
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

Automatic notching options affect both the top and the bottom flange.

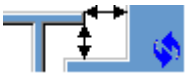
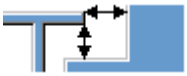
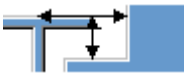
Notch shape

Automatic notching is switched on when you select a notch shape.

Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.

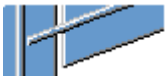
Option	Description
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

Enter the horizontal and vertical values for the cuts.

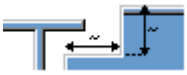
Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.

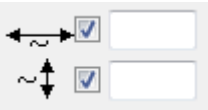
Option	Description
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.



Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

Side of flange notch

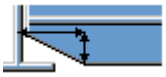
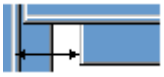
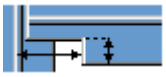
The side of flange notch defines on which side of the beam the notches are created.

Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.

Option	Description
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

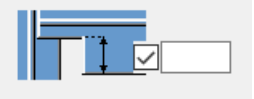
Flange notch shape

The flange notch shape defines the notch shape in the beam flange.

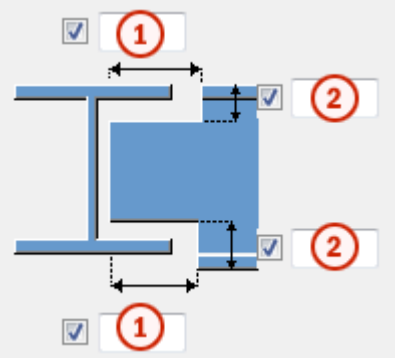
Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

Flange notch depth

Define the flange notch depth.



Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	10 mm
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest 5 mm.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

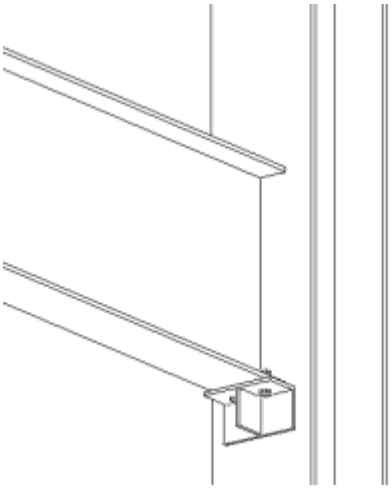
Angle profile box (170)

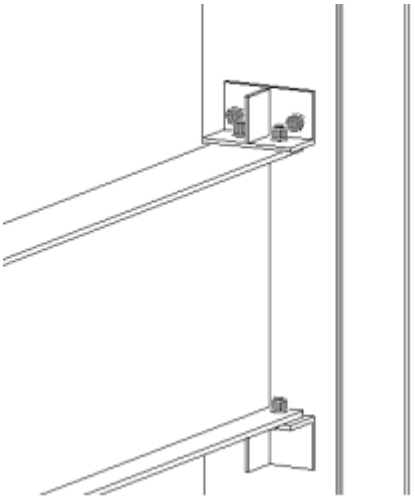
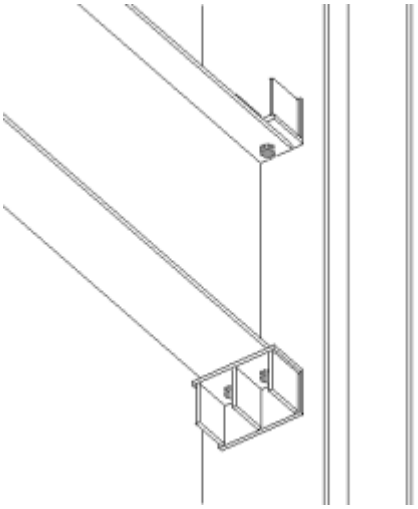
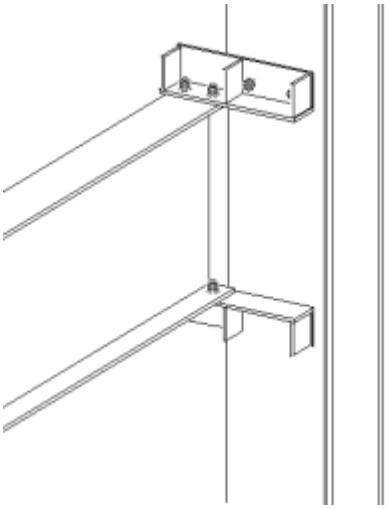
Angle profile box (170) connects a beam to a column with a seat angle. The angles can be placed at the top or bottom, or at both top and bottom flange of the secondary beam. Stiffener plates can be welded to the seat angles. Web stiffening plates can also be added to the secondary beam.

Objects created

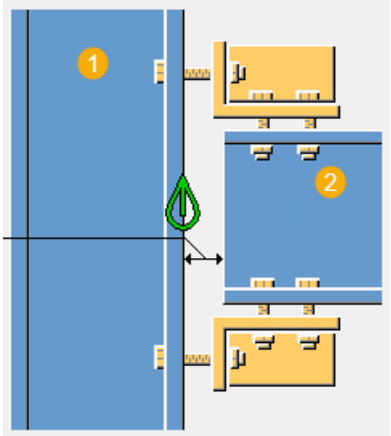
- Clip angles (1 or 2)
- Seat stiffeners (optional)
- Web stiffeners (optional)
- Bolts
- Welds
- Cuts

Use for

Situation	Description
	Beam seat with stiffeners.

Situation	Description
	Beam seat top and bottom with stiffeners. Various bolting options.
	Beam seat. Multiple stiffener options.
	Beam seat. Offset secondary member.

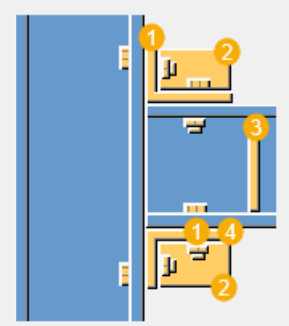
Selection order



1. Select the main part (column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

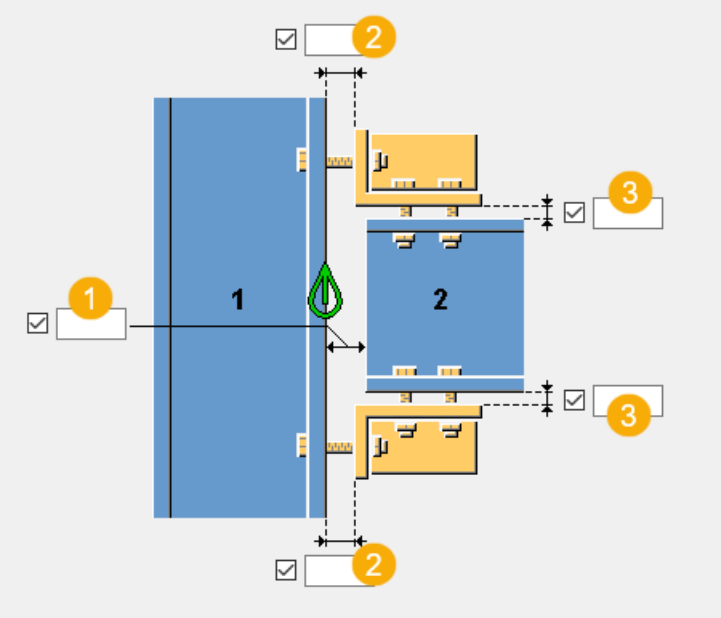


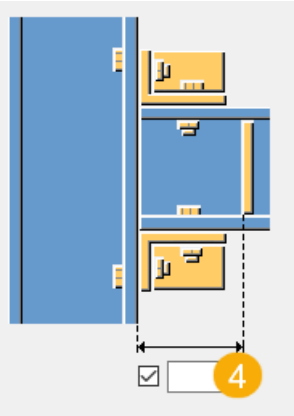
	Part
1	Clip angle
2	Seat stiffener
3	Web stiffener
4	Bottom seat plate

Picture tab

Use the **Picture** tab to control the clearances between the angles and the main and secondary part.

Dimensions





1	Gap between the main part and the secondary part.	`GENERAL / beamedge (0.5”)` `20 mm`
2	Gap between angle profile and the main part.	`0`
3	Gap between angle profile and the secondary part.	`0`
4	Distance of the secondary beam web stiffener from the face of the main part.	

NOTE This information is relevant only to the imperial environment. GENERAL defaults can be found in the `joints.def` file in the system folder and can be modified as required.

Parts tab

Use **Parts** tab to define the stiffeners, clip angle or bottom plate.

Parts

Option	Description
Upper stiffener **Lower stiffener**	Stiffener thickness, width and height. The default values for the height and width are based on the selected profile or bottom plate dimensions. The default stiffener thickness is 10 mm. The default value in the `joints.def` file is `GENERAL/shearplatethk (0.375”).`
Bottom plate	Enter values for thickness and width to create a seat plate instead of a seat angle.
Profile	Select the seat angle profile from the profile catalog. The default angle profile is **L150*100*10** or **L4X4X3/8**. The default value in the `joints.def` file is `GENERAL / lsize`.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

Use the **Parameters** tab to define the seat angle positions. Seat angles carry loads from the secondary part. Seat angles can be positioned to top, bottom or both flanges of the secondary part. The seat angle can be stiffened, and bolted or welded to the main and secondary parts.

Notching

When you create the connection to the main part web, the secondary beam can be fitted to the web and notched by the main part flanges.

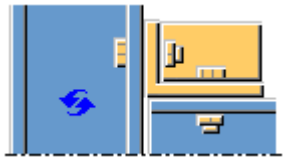
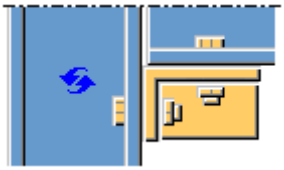
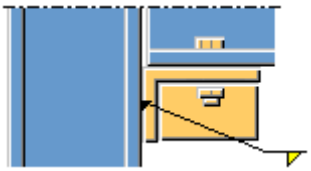
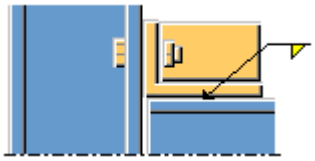
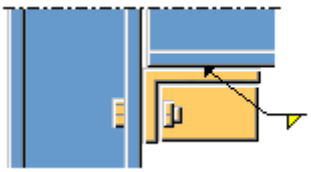
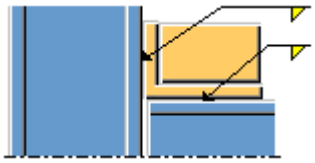
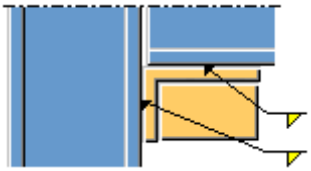
Option	Description
	Default Fitted and notched AutoDefaults can change this option.
	Fitted and notched The secondary part is fitted and notched.
	Fitted The secondary part is fitted, but not notched.
	Notched The secondary part is notched, but not fitted.
	None The secondary part is not fitted or notched.

Seat position

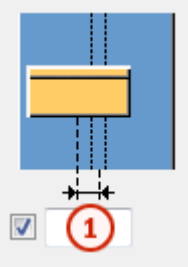
Option	Description
	Default Bottom AutoDefaults can change this option.
	Top Creates a seat at the top of the secondary part.
	Bottom Creates a seat at the bottom of the secondary part.
	Both Creates two seats: one at the top and another at the bottom of the secondary part.

Seat angle attachment

Seat angle is positioned at the top or at the bottom of the secondary part.

Option for top seat angle	Option for bottom seat angle	Description
		Default Bolted Seat angle is bolted to the main part and to the secondary part. AutoDefaults can change this option.
		Bolted Seat angle is bolted to the main part and to the secondary part.
		Welded-bolted Seat angle is welded to the main part and bolted to the secondary part.
		Bolted-welded Seat angle is bolted to the main part and welded to the secondary part.
		Welded Seat angle is welded to the main part and to the secondary part.

Seat angle offset



	Description
1	Seat angle horizontal offset from the center line of the main part.

Seat angle rotation

Option	Description
	Default Seat angle is not rotated. AutoDefaults can change this option.
	Seat angle is not rotated.
	Seat angle is rotated horizontally by 90 degrees. To stiffen the rotated angle, select the **Middle** option in the **Middle stiffener position** list.
	Uses a bottom plate as the seat instead of the angle profile.

Seat angle orientation

Option	Description
	Default The longer leg of the seat angle is connected to the secondary part. AutoDefaults can change this option.
	The longer leg of the seat angle is connected to the secondary part.
	The longer leg of the seat angle is connected to the main part.
	Automatic The longer leg of the seat angle is connected to the part where bolts reach furthest from the seat angle corner.

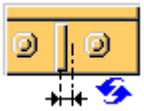
Stiffener type

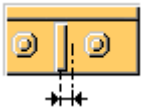
Option	Option	Description
		Default Rectangular stiffener plate AutoDefaults can change this option.
		Rectangular stiffener plate
		Triangular stiffener plate
		The line connecting the ends of the seat angle legs defines the stiffener plate shape.

Side stiffener position

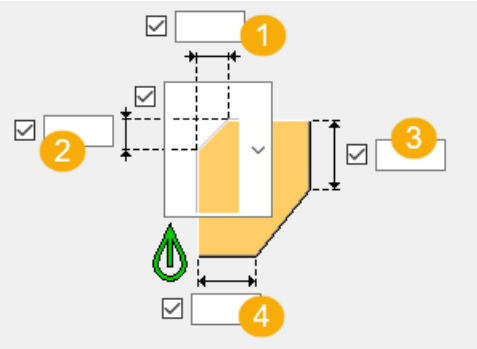
Option	Description
	Default No side stiffeners are created. AutoDefaults can change this option.
	No side stiffeners are created.
	Near side stiffeners are created.
	Far side stiffeners are created.
	Near side and far side stiffeners are created.

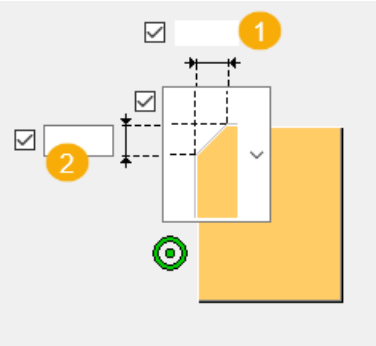
Middle stiffener position

Option	Description
	Default According to bolts AutoDefaults can change this option.

Option	Description
	No middle stiffener plate is created.
	Middle stiffeners The stiffener plate is positioned in the middle of the seat angle. Enter the number of middle stiffeners in the **Number of middle stiffeners** box. Multiple stiffeners are centered and equally spaced.
	According to bolts The stiffener plate is positioned between the bolts in the middle of the bolt spacing. By default, stiffener is created between every two bolts. Enter the number of middle stiffeners in the box below the **According to bolts** option.

Chamfer dimensions





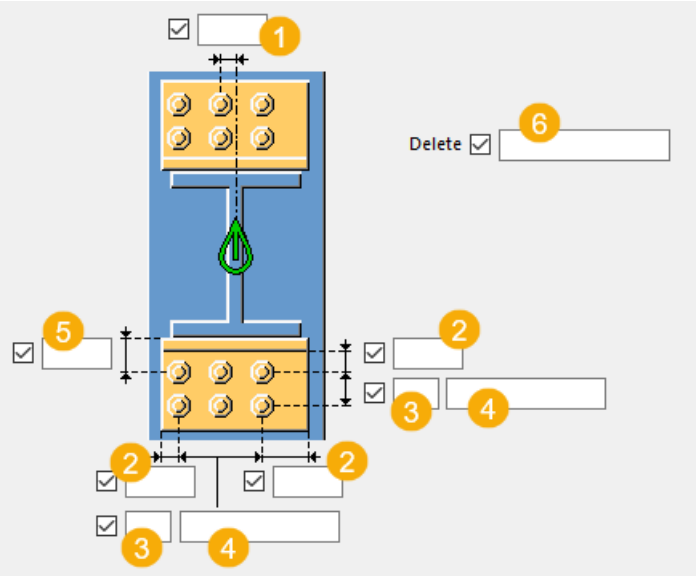
	Description	Default
1	Horizontal dimension of the stiffener plate chamfer.	Equal to angle profile rounding.
2	Vertical dimension of the stiffener plate chamfer.	Equal to angle profile rounding.
3	Vertical dimension of the stiffener plate shape cut line.	
4	Horizontal dimension of the stiffener plate shape cut line.	

Chamfer type

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Pbolts tab

Use the **Pbolts** tab to control the properties of the bolts connecting the clip angle to the main part.



	Description
1	Dimension for horizontal bolt group position. The dimension is defined from the middle line of the secondary beam.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position. The dimension is defined from the bottom of the secondary beam.
6	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

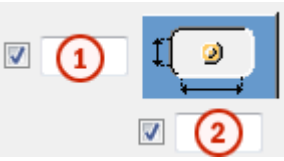
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.

Option	Description	Default
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

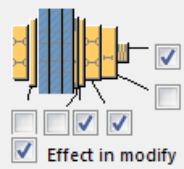


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

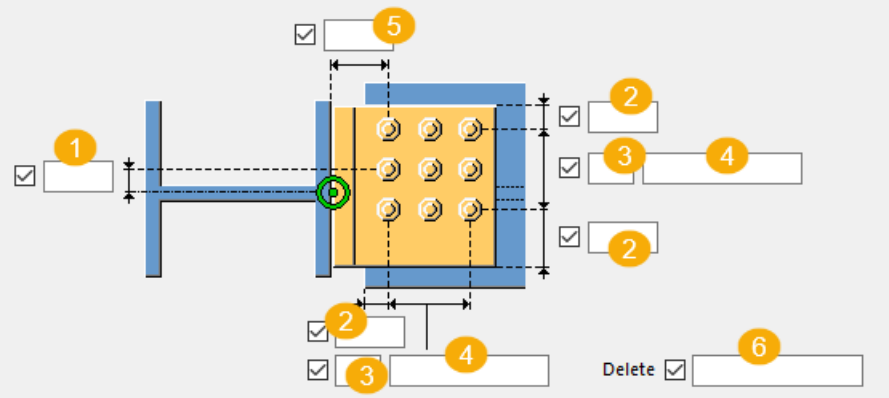
Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Sbolts tab

Use the **Sbolts** tab to control the properties of the bolts connecting the clip angle to the secondary part.



	Description
1	Dimension for vertical bolt group position. The dimension is defined from the middle line of the secondary beam.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for horizontal bolt group position. The dimension is defined from the bottom of the secondary beam.
6	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

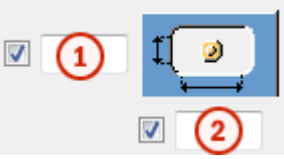
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	

Option	Description	Default
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

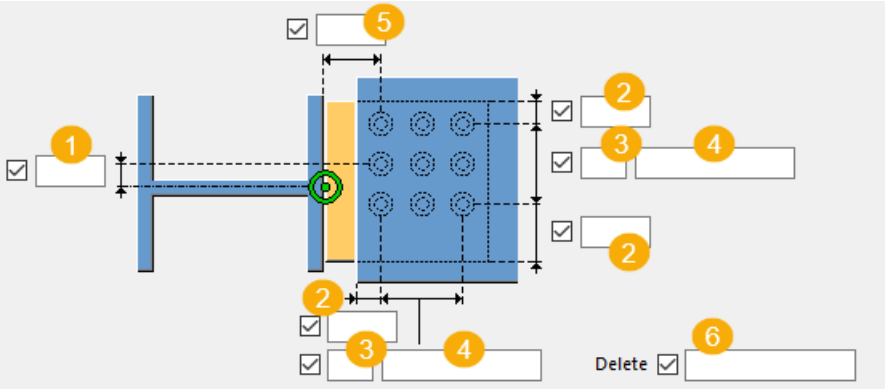
Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

SBoltsDown tab

Use the **SBoltsDown** tab to control the properties of the bolts connecting the bottom seat angle to the secondary part.

Secondary bolt group dimensions



	Description
1	Horizontal bolt group position from the end of the secondary beam.
2	Bolt edge distance.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

	Description
5	Vertical bolt group position. Reference point is from the bottom of the secondary beam.
6	Deletes bolts from the bolt group. Enter the bolt numbers of the bolts to delete, separated by a space. Bolt numbers run left to right and top down.

Bolt basic properties

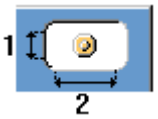
Option	Description	Default

Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
		`joints.def: GENERAL / boltdia`
		`16 mm (0.75”)`

Bolt standard	The bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	The gap between the bolt and the hole.	
Thread in mat	Defines whether or not the thread may be within the bolted parts when using bolts with a shaft. This has no effect when using full-threaded bolts.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

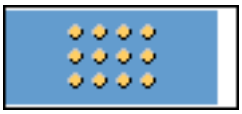
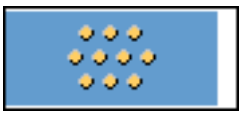
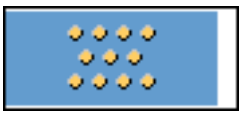
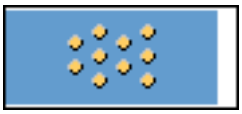
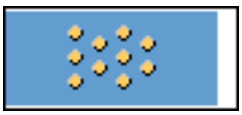
You can define slotted, oversized, or tapped holes with the following options.



Option	Description	Default
1	Slotted hole X dimension or allowance for oversized holes.	`0`, which results in a round hole.
2	Slotted hole Y dimension.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Member(s) in which slotted holes are created. The options depend on the component.	

Staggering of bolts

Option	Description
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beams and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

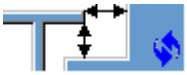
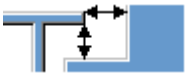
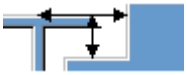
Automatic notching options affect both the top and the bottom flange.

Notch shape

Automatic notching is switched on when you select a notch shape.

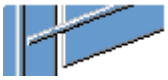
Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

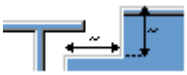
Enter the horizontal and vertical values for the cuts.

Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.

Notch position

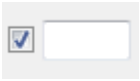
Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.

Option	Description
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.



Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

Side of flange notch

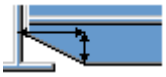
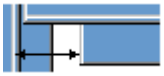
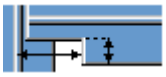
The side of flange notch defines on which side of the beam the notches are created.

Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.

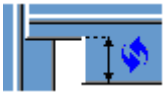
Option	Description
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

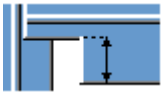
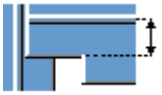
Flange notch shape

The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

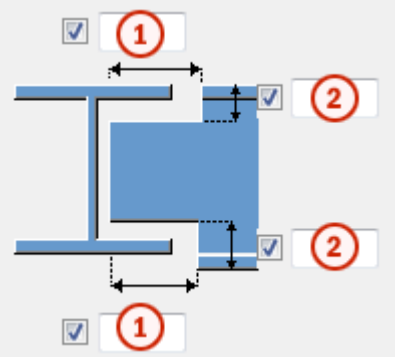
Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.

Option	Description
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.

Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	`10 mm`
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest `5 mm`.

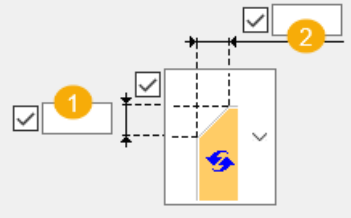
Stiffeners tab

Use the **Stiffeners** tab to control the properties of the near side and far side stiffeners created in the secondary part web.

Option	Description
Stiffener NS **Stiffener FS**	Stiffener thickness, width and height.

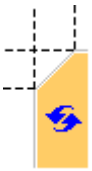
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Chamfer dimensions



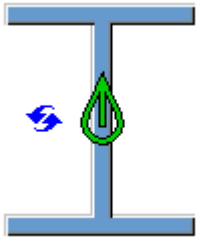
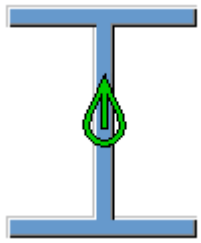
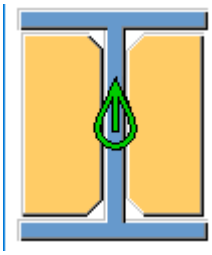
	Description	Default
1	Vertical dimension of the chamfer.	10 mm
2	Horizontal dimension of the chamfer.	10 mm

Chamfer type

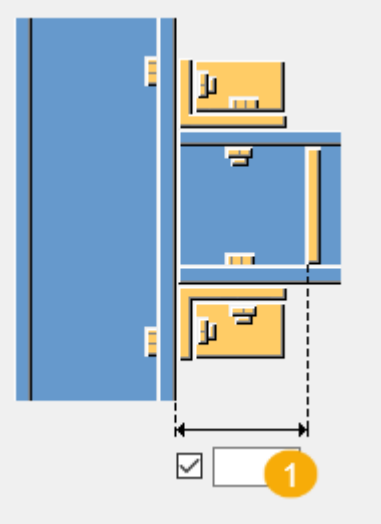
Option	Description
	Default. Line chamfer AutoDefaults can change this option.
	No chamfer

Option	Description
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Stiffener creation

Option	Description
	Default Stiffeners are not created. AutoDefaults can change this option.
	Stiffeners are not created.
	Stiffeners are created.

Stiffener position



	Description
1	Stiffener edge distance from the main part flange.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

5.8 Opening connections

This section introduces components that can be used in steel openings.

Click the links below to find out more:

- Create hole around part (92) (page 1872)
- Rebar hole (page 1877)
- Notch (76) (page 1880)

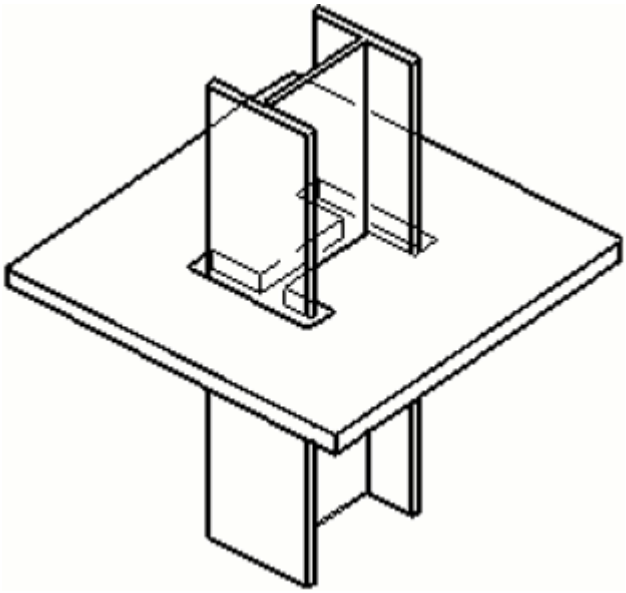
Create hole around part (92)

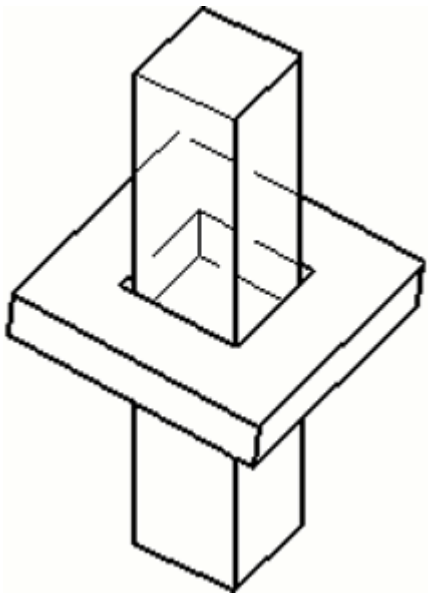
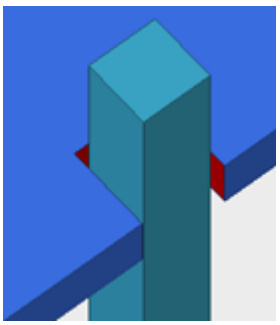
Create hole around part (92) cuts a hole to a part using another part. The part that cuts the hole is perpendicular to the part that is cut.

Objects created

- Cuts

Use for

Situation	Description
	Steel column creates a cut through a plate.

Situation	Description
	Concrete column creates a cut through a plate.
	Concrete column creates a cut in a plate.

Selection order

1. Select the main part (column).
2. Select the secondary part.

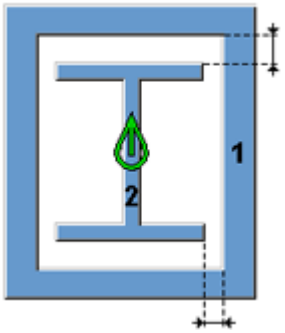
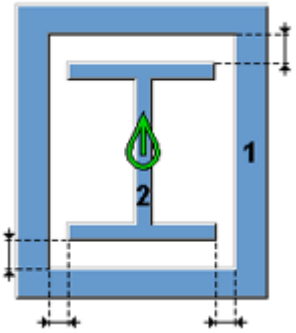
   The connection is created automatically when the secondary part is selected.

Picture tab

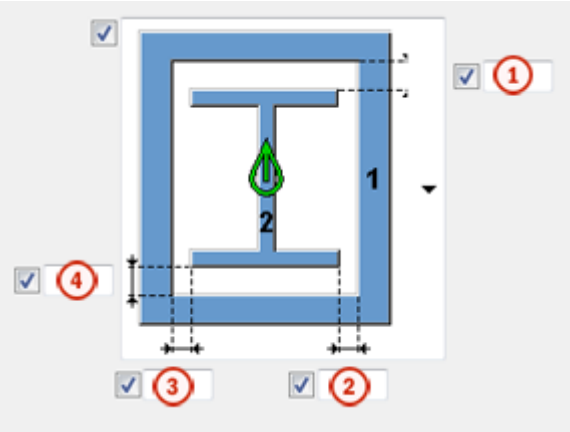
Use the **Picture** tab to control the gap dimensions and whether the dimensions are the same on both sides of the gap.

Gap side

Define whether the gap dimensions are the same on both sides of the gap.

Option	Description
	Gap dimensions are the same on both sides.
	Gap dimensions are different on each side.

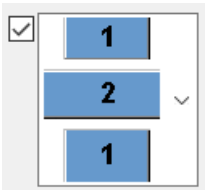
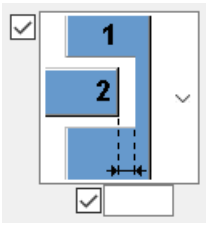
Gap dimensions



	Description
1	Gap between the column and the secondary part in the vertical direction.
2	Gap between the column and the secondary part in the horizontal direction.
3	Gap between the column and the secondary part in the horizontal direction. To define this dimension, select the option that the gap is different on each side.

	Description
4	Gap between the column and the secondary part in the vertical direction. To define this dimension, select the option that the gap is different on each side.

Part cut

Option	Description
	A cut is always created through the whole main part.
	Define the depth of the cut from the edge of the secondary part.

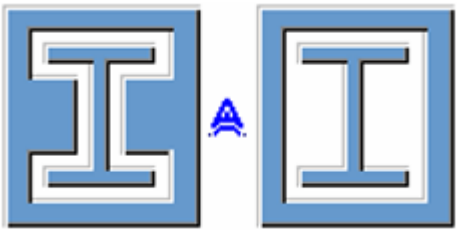
Parameters tab

Use the **Parameters** tab to control the cut properties and type of the cut.

Cut properties

Option	Description
Max. rectangle size	Define the maximum size of a rectangle cut.
Cut perpendicular to main part	Define whether the cut is perpendicular to the main part. This option works for rectangular cuts.
Cut part name	Define a name for the cut part.

Cut type

Option	Description
	Default Creates an exact or a rectangular cut. AutoDefaults can change this option.
	Rectangular Creates a rectangular cut using the coordinates of the cutting part edges.
	Exact Creates a cut along the edges of the cutting part. The cutting part has to be perpendicular to the part it cuts. You can use this option for I, round and tube profiles.
	Automatic This option selects the cut type depending on the cut size defined in **Max. rectangle size**. If the size of the cutting part is larger than the size defined in **Max. rectangle size**, an exact cut is created. Otherwise, a rectangular cut is created.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

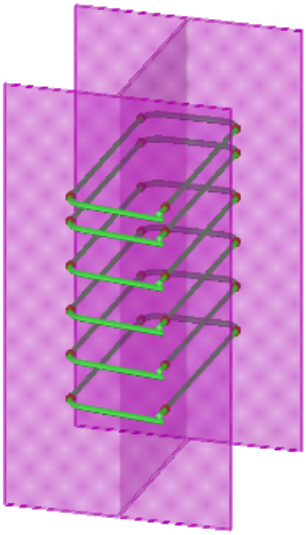
Rebar hole

Rebar hole creates holes for reinforcing bars in a steel column, beam, or contour plate.

Objects created

- Holes
- Cuts

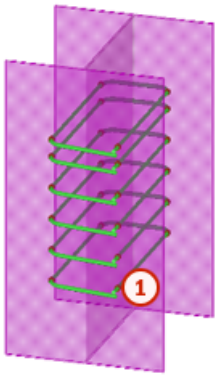
Use for

Situation	Description
	Holes created for reinforcing bars in a steel beam.

Selection order

1. Select the main part (column, beam, or contour plate).
2. Select the secondary part (reinforcing bar group).
3. Click the middle mouse button to create the holes for the reinforcing bars.

Part identification key



	Part
1	Hole for reinforcing bar

Parameters tab

Use the **Parameters** tab to control the properties of the reinforcing bar holes.

Reinforcing bar hole properties

Option	Description	Default
Bolt standard	Select the bolt standard: • **6914** • **7968** • **7990** • **ASS 1** • **ASS 2** • **UNDEFINED_BOLT**	`6914`
Rounding type	Select the rounding type: • **None** The rounding value is the reinforcing bar diameter + hole tolerance. • **Round off** The rounding value is the nearest integer number divisible by the rounding precision value.	No default value Example dimensions: • Reinforcing bar diameter = `21.6 mm` • Hole tolerance = `3 mm` • Rounding precision = `2 mm` • None, rounding = `24.6 mm`

Option	Description	Default
	• **Round up** The rounding value is the next integer number divisible by the rounding precision value. • **Round down** The rounding value is the previous integer number divisible by the rounding precision value. • **Round by table** Define the reinforcing bar diameter, hole diameter, and the slotted hole extension.	• Round off, rounding = 24 mm • Round up, rounding = 26 mm • Round by own, rounding = 24 mm
Hole tolerance	Define the hole tolerance. When you select **Round by table** as the rounding type, you cannot define the hole tolerance.	0 mm
Rounding precision	Define the rounding precision. When you select **Round by table** as the rounding type, you cannot define the rounding precision.	1 mm
Dimension table	Define the reinforcing bar diameter, hole diameter, and slotted hole extension. To define the dimensions in the table, select the **Round by table** option as the rounding type. **Rebar hole** uses an existing bolt size and adjusts the hole tolerance value until the desired **Hole diameter** is reached. Use the **+** and the **-** buttons to add and delete rows from the table.	

Advanced tab

Use the **Advanced** tab to control the hole type, vertical offset, and angle range dimension.

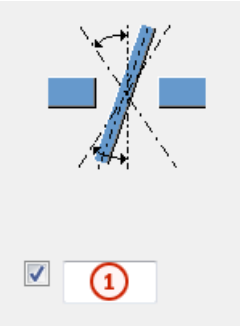
Hole type

Option	Description	Default
Hole type	Select the hole type: • **Bolt hole** • **Part cut** • **Bolt hole + part cut**	`Bolt hole`

Vertical offset

Option	Description
	Default The hole is not offset.
	The hole is offset upward to directly support the reinforcing bar to keep the bar in the correct vertical position. The hole can be offset if the hole is a circular hole.

Angle range dimension



	Description	Default
1	Angle range dimension. A round hole is created within the defined range. Holes that go beyond the range are transformed into slotted holes.	`5 mm`

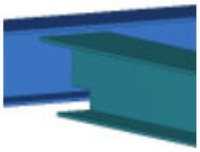
Notch (76)

Notch (76) creates a notch to the secondary beam.

Objects created

- Cuts

Use for

Situation	Description
	Secondary beam is notched.

Selection order

1. Select the main part.
2. Select the secondary part.

   The connection is created automatically when the secondary part is selected.

Notch tab

Use the **Notch** tab to create notches to the secondary beams and to control the notch properties.

Automatic notching

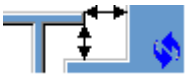
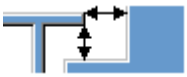
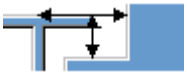
Automatic notching options affect both the top and the bottom flange.

Notch shape

Automatic notching is switched on when you select a notch shape.

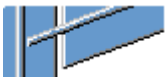
Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

Enter the horizontal and vertical values for the cuts.

Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.

Option	Description
	Creates the cut above the main beam flange.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

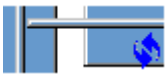
Side of flange notch

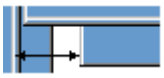
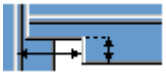
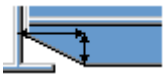
The side of flange notch defines on which side of the beam the notches are created.

Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

Flange notch

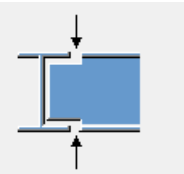
The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Creates cuts to the flange with default values unless you define other values.

Option	Description
	Creates cuts to the flange according to the value you define to make it flush with the web.
	Creates cuts to the flange according to the values you define.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	The flange is not cut.

Cut dimensions

Define the vertical flange cut dimensions.



Parameters tab

Use the **Parameters** tab to define the notch type and the clearance between the parts.

Option	Description
Fitting type	Select the type of notch created between the main part and the secondary part.
Clearance	Define the clearance value for the gap created between the parts.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

5.9 Bracing

This section introduces components that can be used in steel bracing structures.

- Tensioner (7) (page 1885)
- Tensioner brace (13) (page 1904)
- Tensioner brace and compression bar (13) (page 1913)
- Tensioner central gusset (18) (page 1926)
- Turnbuckle bracing (S3) (page 1944)
- Muffe (26) (page 1952)
- Turnbuckle connection (126) (page 1955)
- Generation of purlins (50) (page 1959)
- Gusset+ T (page 1972)

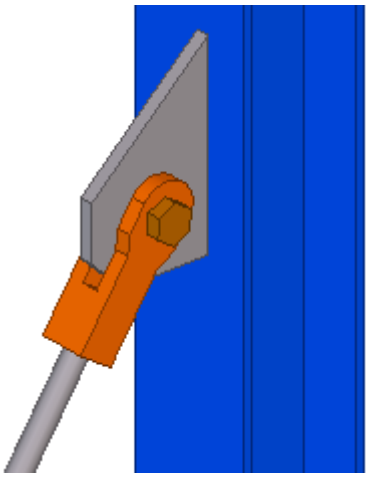
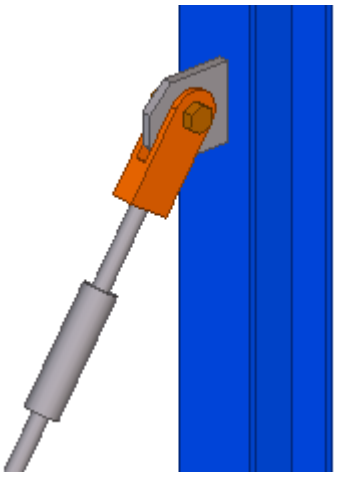
Tensioner (7)

Tensioner (7) connects a column or a beam to a brace with a forked plate or a flat plate. Optionally, a gusset plate can be created.

Objects created

- Gusset plate (optional)
- Forked or flat plate
- Tensioner (optional)
- End plate (optional)
- Bolts
- Welds

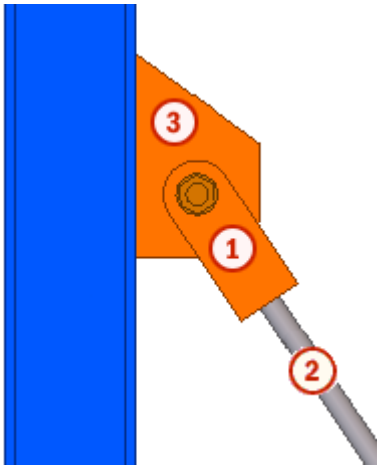
Use for

Situation	Description
	Forked plate is welded to a bracing rod and bolted to a gusset plate. The gusset plate is welded to the main part.
	Simplified tensioner in the bracing rod.

Selection order

1. Select the main part (column or beam).
2. Select the secondary part(s) (brace).
3. Click the middle mouse button to create the component.

Part identification key

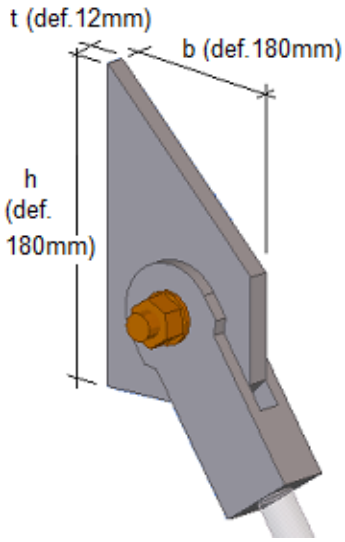


	Part
1	Connection plate (forked plate)
2	Bracing rod
3	Gusset plate

Plate tab

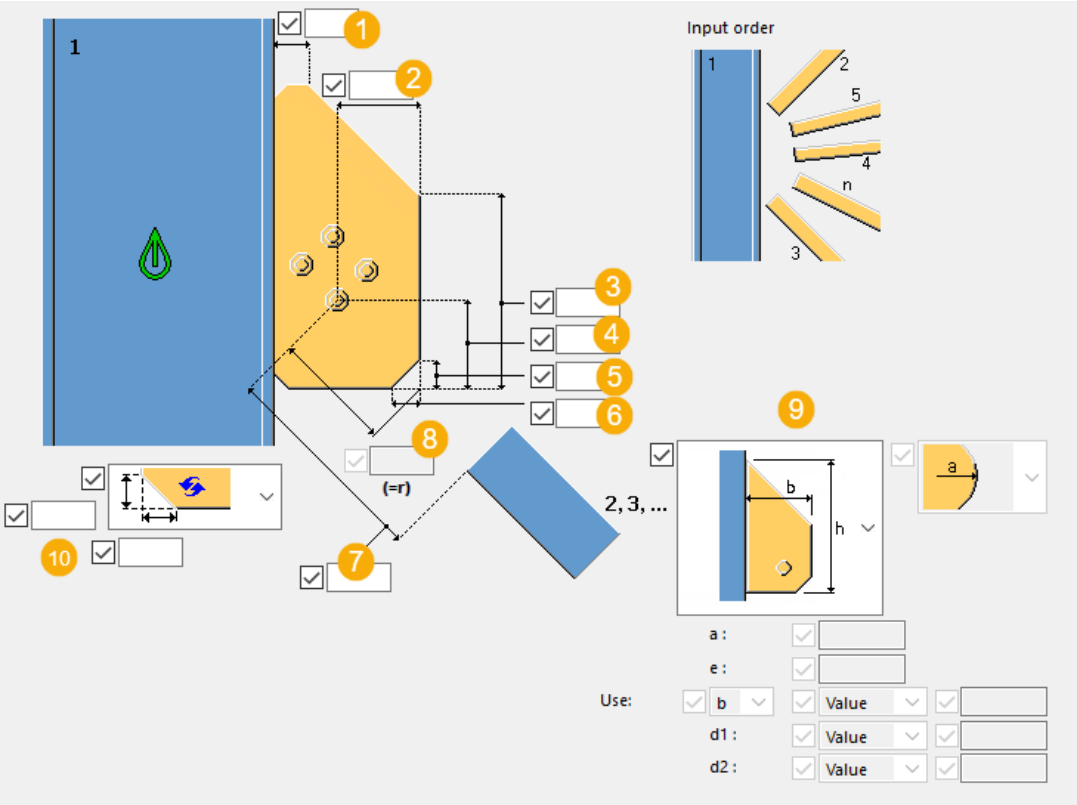
Use the **Plate** tab to control the gusset plate dimensions and shape.

Plate

Part	Description	Default
Gusset plate	Define the gusset plate thickness, width and height. 	12 mm 100 mm 180 mm

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Gusset plate shape and dimensions



	Description	Default
1	Horizontal top dimension from the column web.	
2	Horizontal bolt edge distance.	50 mm
3	Height of the gusset plate.	80 mm
4	Bottom vertical bolt edge distance.	50 mm
5	Bottom vertical chamfer dimension.	20 mm
6	Bottom horizontal chamfer dimension.	20 mm
7	Bolt edge distance to the bracing rod.	110 mm
8	Bolt edge distance of the chamfer (radius).	
9	Gusset plate shape. Select the plate shape and define the plate dimensions.	
10	Select the chamfer shape. Define the horizontal and vertical chamfer dimensions.	

Fork tab

Use the **Fork** tab to control the size, position, number, orientation and shape of the forked plate.

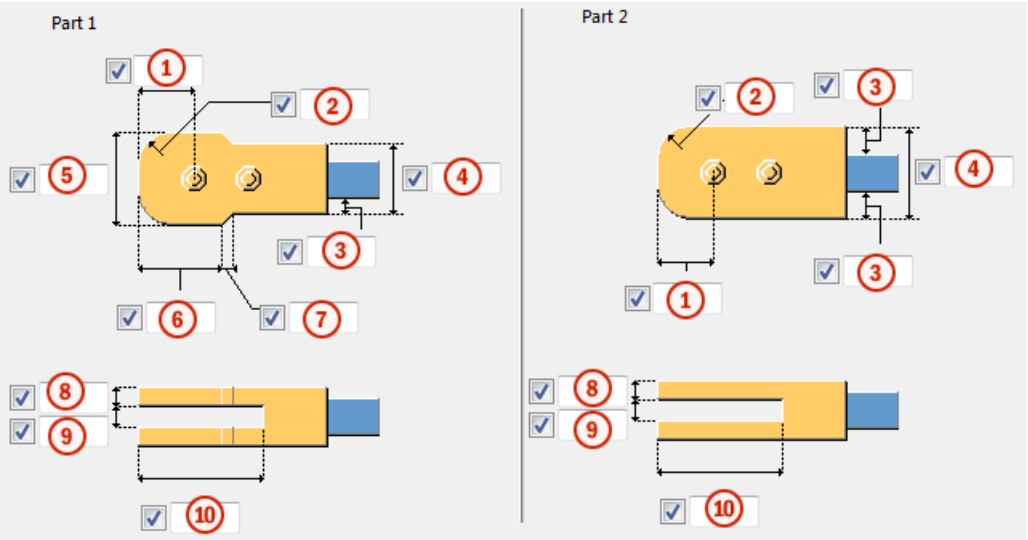
Forked plate

Part	Description
Forked plate	Select the shape for the forked plate: • **Part 1** shape creates a plate with a circular part. • **Part 2** shape creates a simple plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part**

Option	Description	Default
		material box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Forked plate dimensions

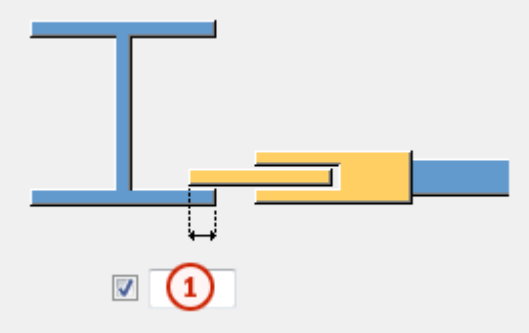


	Description
1	Horizontal bolt edge distance.
2	Chamfer radius of the forked plate.
3	Width of the extension part.
4	Width of the forked plate.
5	Width of the forked plate.
6	Length of the circular part of the forked plate.
7	Chamfer width of the forked plate.
8	Finger thickness of the forked plate.
9	Gap between the fingers of the forked plate.
10	Finger length in the forked plate.

Parameters tab

Use the **Parameters** tab to control the overlap of the gusset plate and the size, position, number and shape of the end plate.

Gusset plate overlap



	Description
1	Define how much the gusset plate overlaps with the main part flange. If you do not enter any value, the gusset plate reaches the main part web.

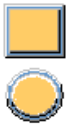
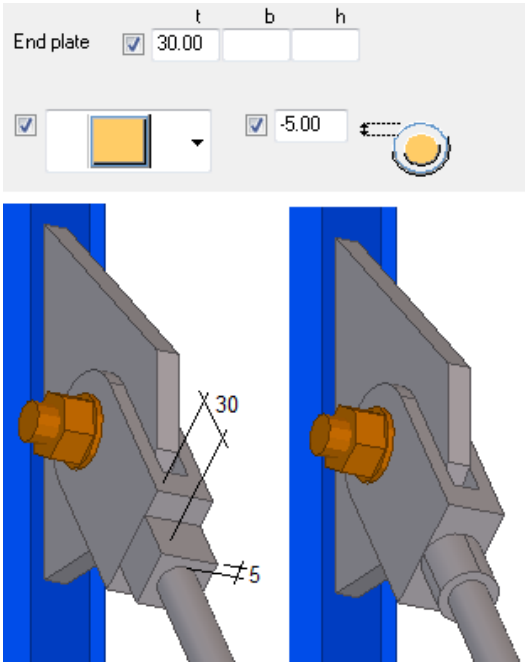
End plate

1	Define the end plate thickness, width, and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Option	Description	Default
Finish	Describes how the part surface has been treated.	

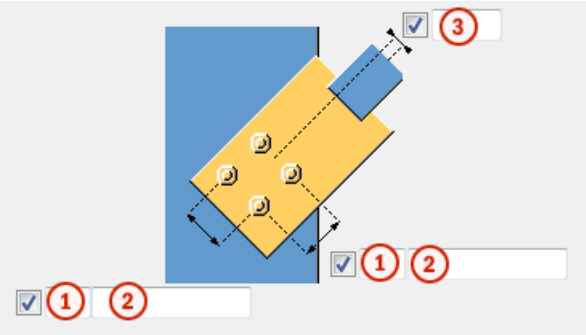
End plate shape

Option	Description
	Select the end plate shape.
	Define the end plate offset from the bracing rod.

Bolts tab

Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions



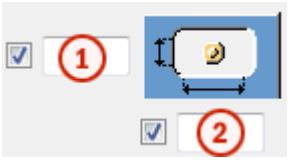
	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Define the bolt offset from the center line of the bracing rod.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



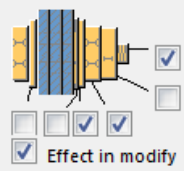
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Tensioner tab

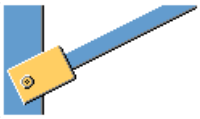
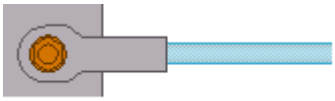
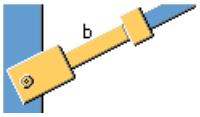
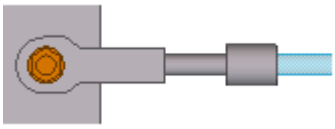
Use the **Tensioner** tab to add a tensioner, to control the bracing levels and bracing offsets.

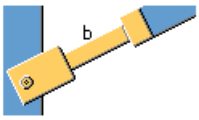
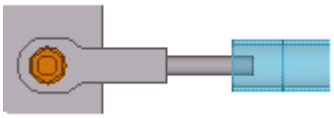
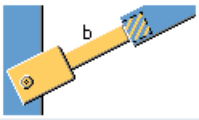
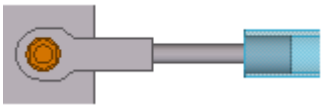
Part

Part	Description	Default
Tensioner T	Define the tensioner profile by selecting it from the profile catalog.	D40
Part B	If you have created a tensioner, define the extra windbracing profile by selecting it from the profile catalog.	

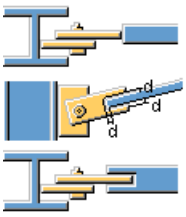
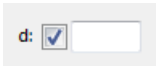
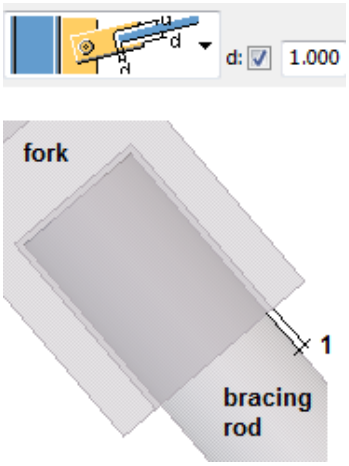
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Tensioner

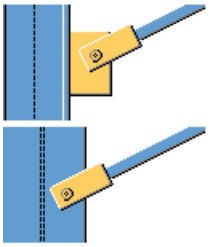
Option	Description	Example
	Tensioner is not created.	
	Tensioner is created.	

Option	Description	Example
	Tensioner is created. Stopper part is added to the bracing. Used for compression tubes.	
	Tensioner is created. Stopper part is placed inside the compression tube.	

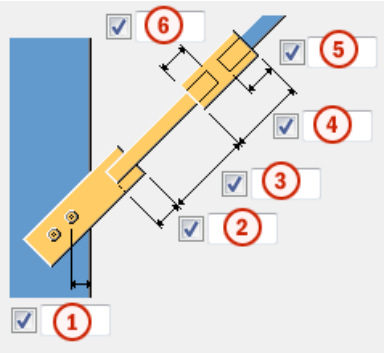
Opening in forked plate

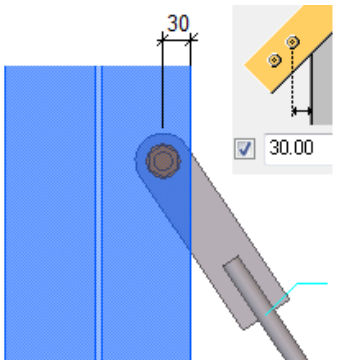
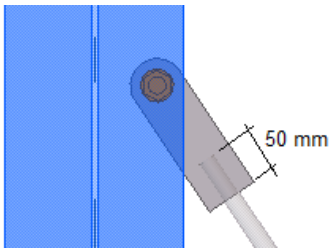
Option	Description	Example
	Select whether an opening is created in the forked plate. The opening is always square. You can define the opening if both the tensioner and extra windbracing are created.	
	Define the gap for the opening. The default value is `1 mm`.	

Gusset plate

Option	Description
	Select whether a gusset plate is created. If you do not select a gusset plate, only the forked plate will be created.

Bracing dimensions



	Description	Example/Default
1	Bolt edge distance from the main part flange when there is no gusset plate. The default value is `30 mm`.	
2	Bracing rod overlap.	
3	Length of the extra windbracing between the forked plate and the tensioner.	The default value is `300 mm`.
4	Length of the tensioner.	The default value is `40 mm`.

	Description	Example/Default
5	Bracing overlap in the tensioner.	The default value is 0 mm.
6	Extra windbracing overlap in the tensioner.	The default value is 0 mm.

Bracing position

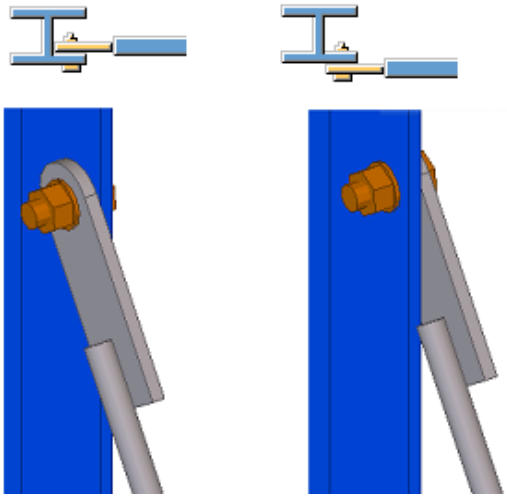
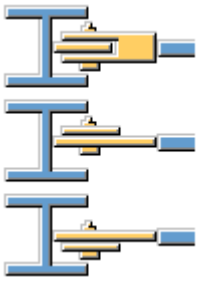
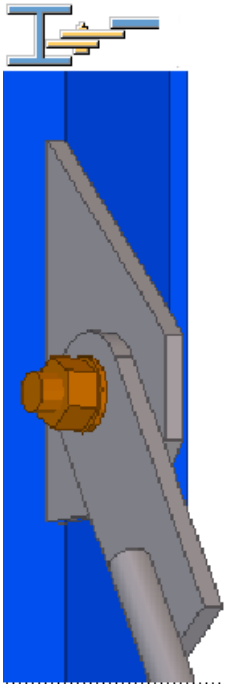
Option	Description
	Position of the bracing on the main part flange. This option is useful especially if there is no gusset plate.

Plate position

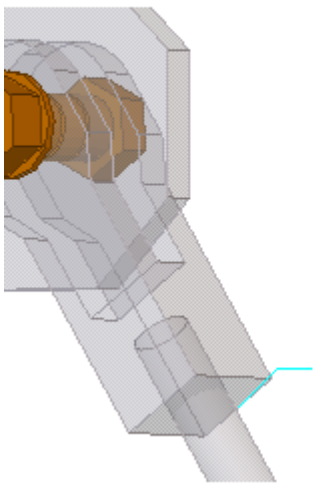
Option	Description
	Select the position of the forked plate on the main part web.

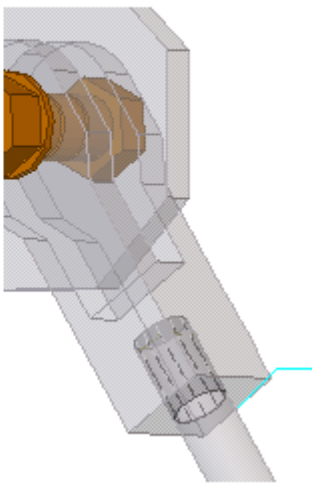
Forked plate position on bracing

Option	Descripton	Example
	Select the position of the forked plate on the bracing. This option is useful especially with flat plates.	

Cut part B in fork

Define whether the forked part is cut if the bracing rod goes through the forked part. The forked part cut adapts to the bracing rod size.

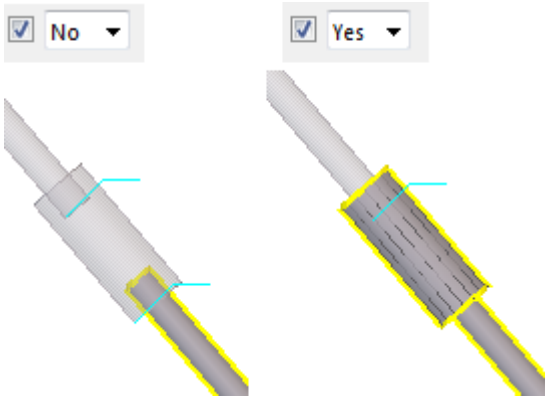




Tensioner T add to secondary

Select whether the tensioner is added to the secondary part or handled as a loose part and welded to the bracing rod.

- **Yes** adds the tensioner to the bracing rod.
- **No** welds the tensioner to the bracing rod as a loose part.

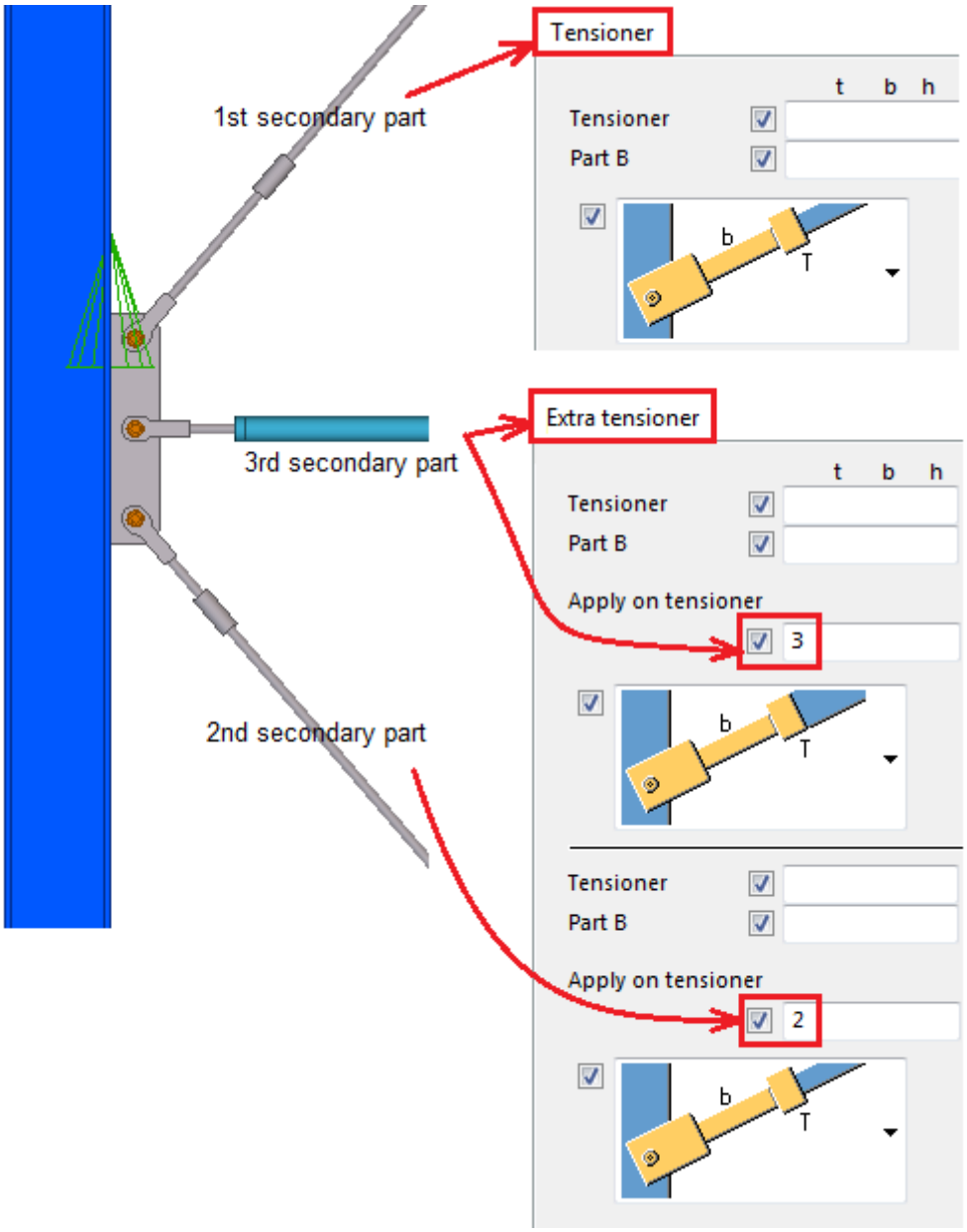


Extra tensioners tab

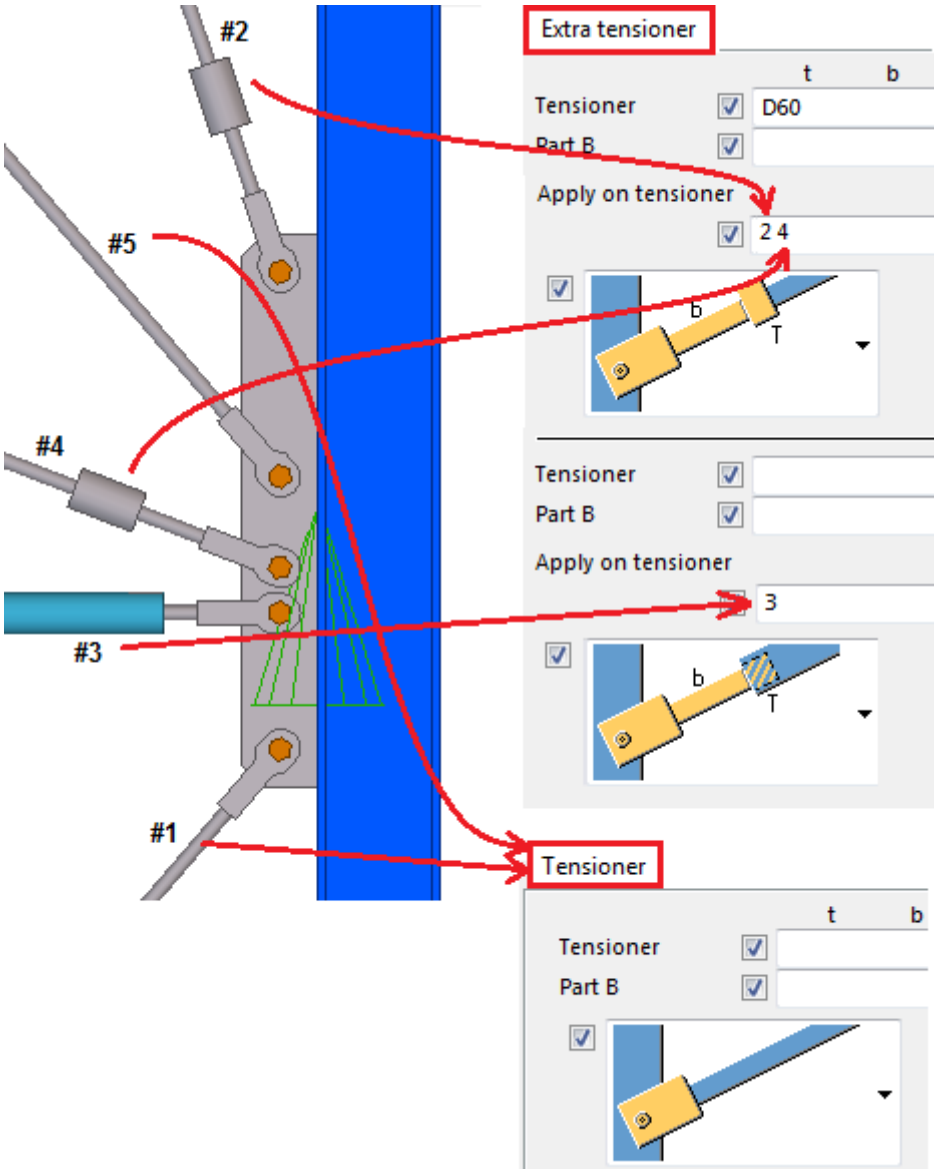
Use the **Extra tensioners** tab to add extra tensioners. Two different tensioner types can be defined.

Extra tensioners

If there is one bracing, define the tensioner on the **Tensioner** tab. If there are more bracings, define the tensioners for the second, third, etc. bracing on the **Extra tensioners** tab. Define the bracing numbers in the **Apply on tensioner number** box.



The tensioners whose numbers are not entered are created with the properties defined on the **Tensioner** tab.



For instructions on tensioner bracing dimensions, **Cut part B in fork** and **Tensioner T add to secondary**, see the instructions on the **Tensioner** tab.

Parts

Part	Description
Tensioner T	Define the tensioner profile by selecting it from the profile catalog.
Part B	If you have created a tensioner, define the extra windbracing profile by selecting it from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

UDA tab

Use the **UDA** tab to add information in the user-defined attributes (UDAs) of the parts.

You can define UDAs for the plate and the fork. UDAs can be displayed in drawings and reports.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

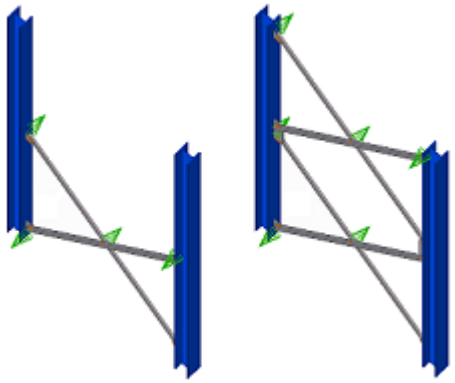
Tensioner brace (13)

Tensioner brace (13) creates one or two bracing crosses between two columns or beams. It is also possible to add connections between columns or beams and the bracing crosses. You can define which connections are used.

Objects created

- Bracing cross (1 or 2)
- Connections between columns or beams and bracing crosses
- Connections in bracing crosses

Use for

Situation	Description
	One or two bracing crosses between two columns.

NOTE To use **Tensioner brace (13)** you need to set the **Up direction** on the **General** tab to a fixed direction: `-x`,`+x`,`-y`,`+y`,`-z`, or `+z`.

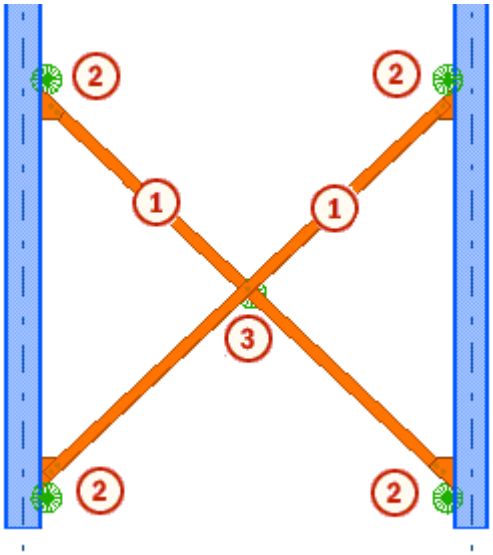
The **Auto** option does not work.

Selection order

1. Select the first main part (column or beam).
2. Select the second main part (column or beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

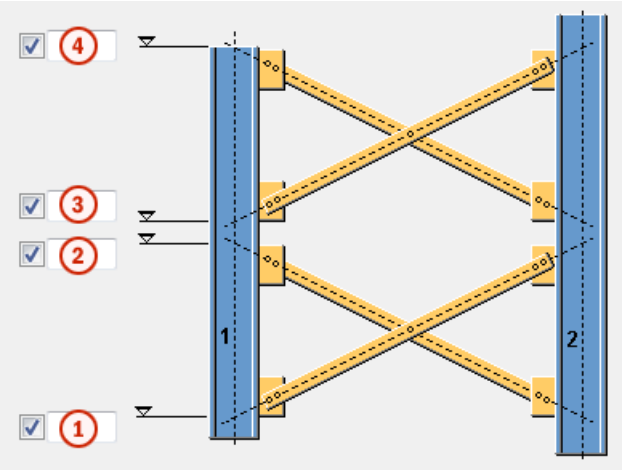


1	Diagonal bracing
2	Connection between the main part and the bracing
3	Connection in the bracing cross

Picture tab

Use the **Picture** tab to control the bracing levels and bracing offsets.

Bracing levels



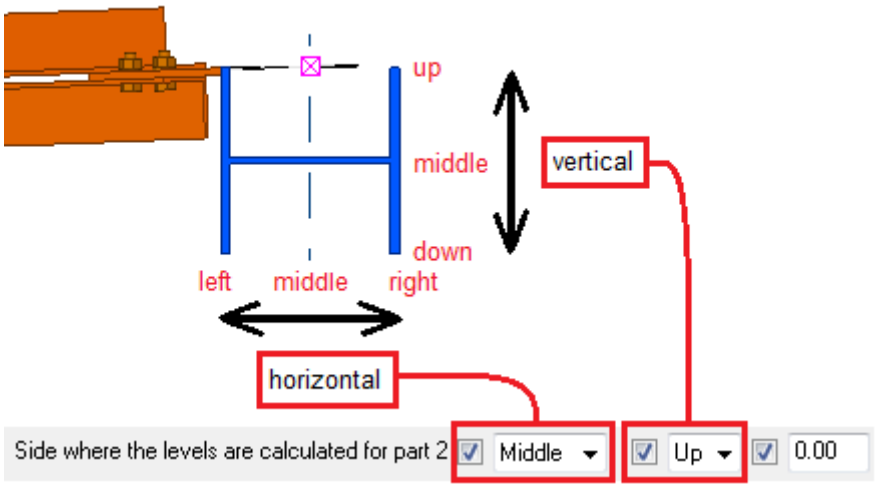
	Description
1	Bottom level of the lower bracing cross.

	Description
2	Top level of the lower bracing cross.
3	Bottom level of the upper bracing cross.
4	Top level of the upper bracing cross.

Bracing reference

For both main parts, define the reference side of the bracing levels. The reference side can be set for both in the horizontal and the vertical direction.

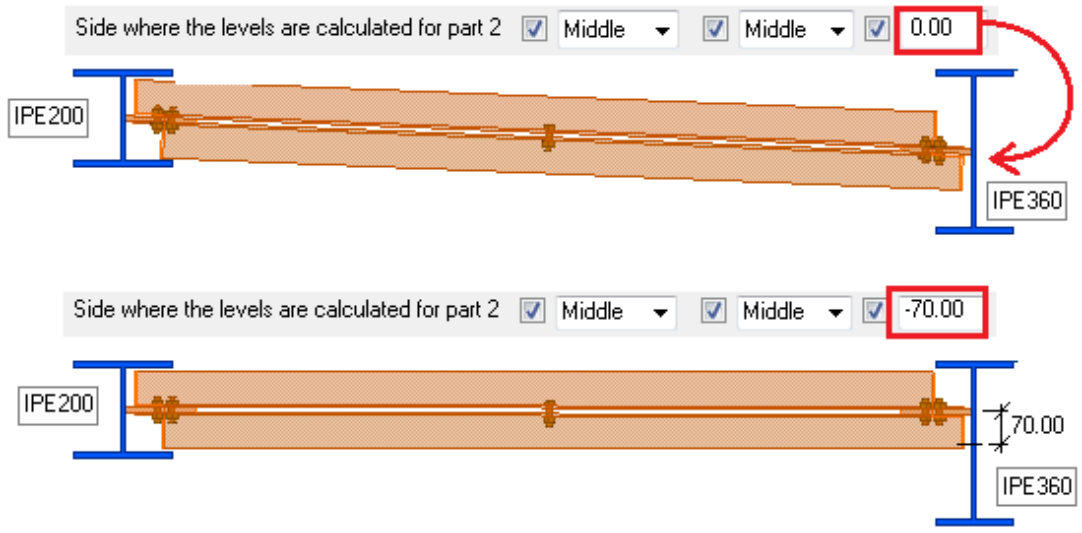
Example:



Bracing offset

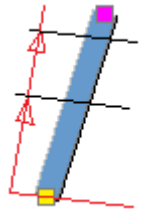
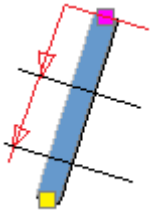
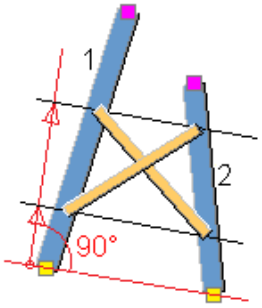
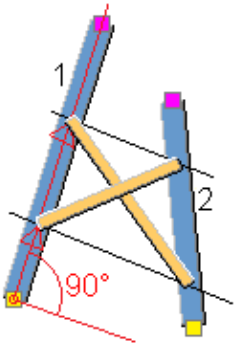
Define the offset perpendicular to the bracing. You can move the created plate or part by entering a value in the x-, y, or z-direction.

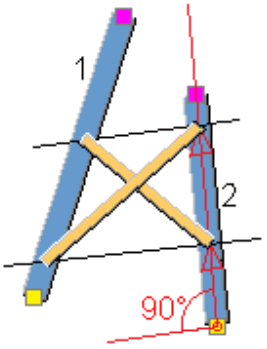
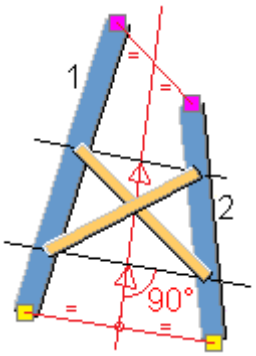
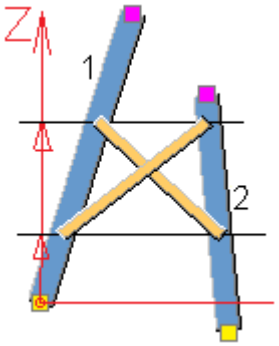
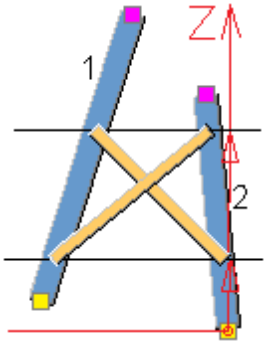
Example:



Levels tab

Use the **Levels** tab to control the bracing direction when the columns are not parallel.

Option	Description		
Direction	Select the direction of the bracing.		Start point of the main part as the reference point.
			End point of the main part as the reference point.
Calculation of levels	Select the reference line used for positioning the bracing when the main parts are not parallel.		Offset from the start point in part 1, perpendicular to the line through the start points.
			Offset from the start point in part 1, in local x-direction.

Option	Description		
			Offset from the start point in part 2, in local x-direction.
			Reference line through the start and end points of the main parts, offset from the start of the reference line.
			Offset from the start point in part 1, in z-direction.
			Offset from the start point in part 2, in z-direction.

Parts tab

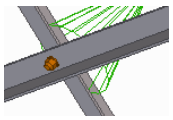
Use the **Parts** tab to control the properties, bracing position, and rotation. Additionally, you can define bracing splitting and shortening values.

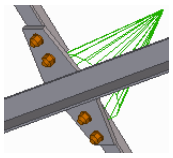
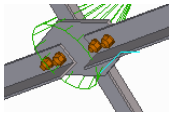
<table>
<tr><th>Option</th><th colspan="3">Description</th></tr>
<tr><td>Windbracing</td><td colspan="3">Define the bracing profile by selecting it from the profile catalog.</td></tr>
<tr><td>Windbracing rotation</td><td colspan="3">Select the rotation for the first and the second bracing element.

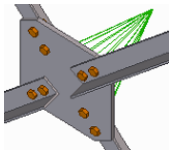
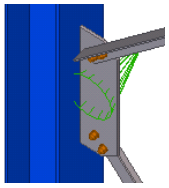
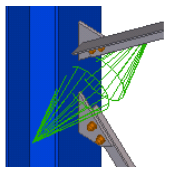
This option is useful when the bracing elements are crossing and they are connected in the crossing.</td></tr>
<tr><td>Windbracing translation</td><td colspan="3">Select the offset of the first and the second bracing element from the reference points.

This option is useful when the bracing elements are positioned so that the first bracing is alongside the second bracing. Typically, the first bracing element is set to Forwards and the second bracing element to Backwards.</td></tr>
<tr><td rowspan="3">Splitting the diagonal bracing</td><td rowspan="3">Select whether the diagonal bracing elements are split or connected with a component.

Define the connecting component on the Joints tab by typing the number of the component in the Connect diagonals with joint number box.</td><td>
Bracing is not split.

Possible component for connecting the diagonal bracing: Seating (30).</td><td>  </td></tr>
<tr><td>
First diagonal bracing element is split.

Possible component for connecting the diagonal bracing: Bolted gusset (11).</td><td>  </td></tr>
<tr><td> </td><td>  </td></tr>
</table>

Option	Description		
		Second diagonal bracing element is split. Possible component for connecting the diagonal bracing: **Bolted gusset (11)**.	
		 Both diagonal bracing elements are split. Possible component for connecting the diagonal bracing: **Central gusset (169)**.	
Connecting bracing crosses	Select whether the gusset plates of two bracing crosses above each other are connected. Define the connecting component on the **Joints** tab by typing the number of the component in the **Connection number** box.	 Bracing crosses are connected with a gusset connection. Possible component for connecting the diagonal bracing: **Bolted gusset (11)**.	
		 Bracing crosses are not connected. Separate connection is created for each diagonal bracing. Possible component for connecting the diagonal bracing: **Bolted gusset (11)**.	

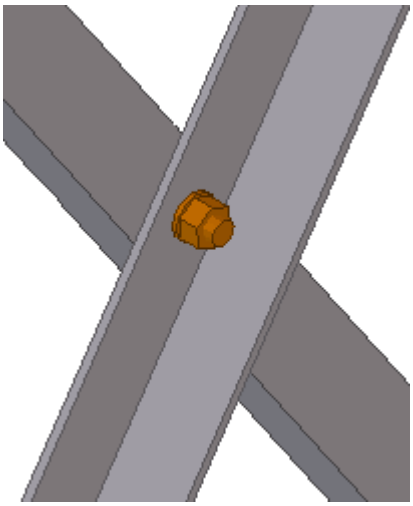
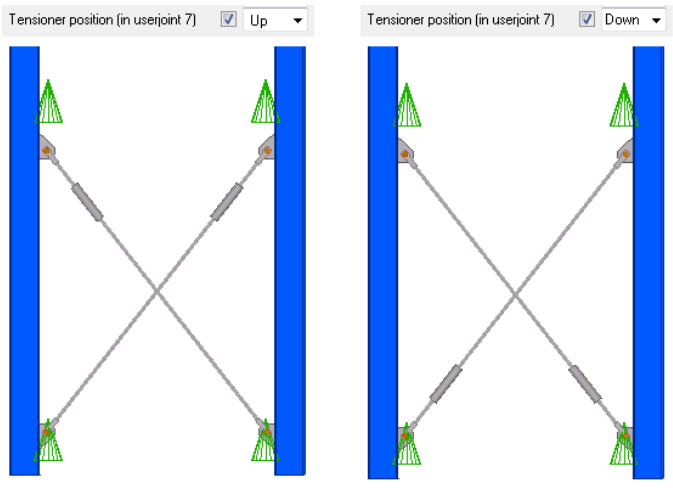
Option	Description
Distance between windbracings	Define the distance between bracing elements. If the bracing elements are crossing each other, this value typically defines the gusset plate thickness.
Shorten windbracings	Define how much the bracing are shortened. The entered value is written in the user-defined attributes of the bracing. The value is used in drawings.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Comment	Add a comment about the part.	

Joints tab

Use the **Joints** tab to define the components used for connecting the columns or beams and the bracing crosses.

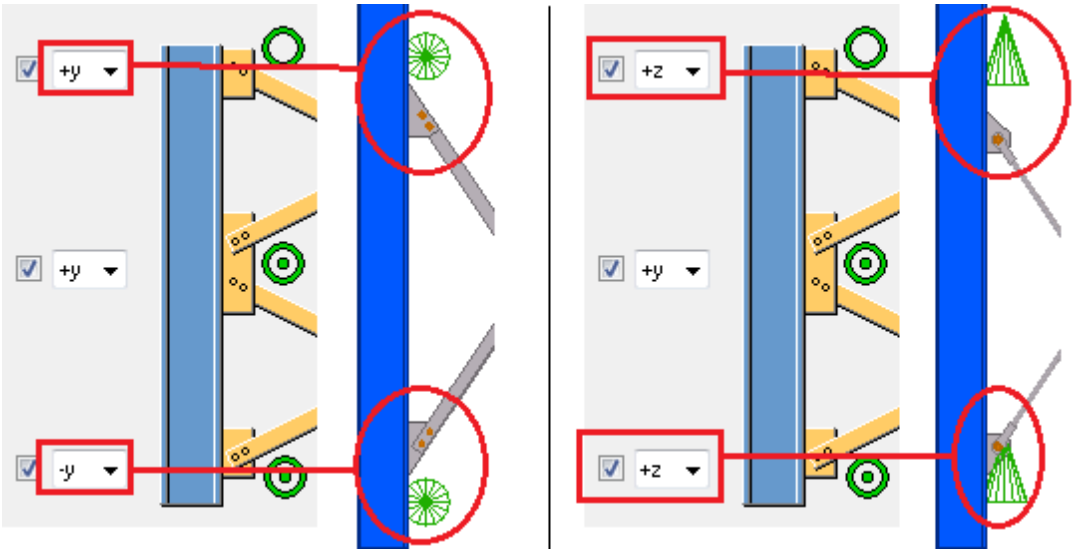
Option	Description
Creation of joints	Select whether connections are created between the bracing elements. **No**: Only the bracing elements are created. **Yes**: Components are added between the bracing elements.
Connection number **Connect diagonals with joint number**	Type the component number that is used for connecting the bracing or the diagonal bracing cross, and the component application number.

Option	Description
Userjoint application number **Joint direction**	Select the connection direction for the diagonal bracing cross. • Default component for the bracing is **Bolted gusset (11)**. • Default component for the diagonal bracing cross is **Seating (30)**. Example: 
Configuration file	Configuration setting for the connection. For example, if you type `CS_M13`, it means that a setting named `CS_M13` must be available for the used connection.
Tensioner position	Define the tensioner position if the **Tensioner (7)** connection is used. Example of **Tensioner (7)** position in both the **Up** and the **Down** position. 

Joints dir tab

Use the **Joints dir** tab to control the up directions of the connections used between the main parts and the diagonal bracing.

In the example below, **Seating (30)** has been defined as the connecting component on the **Joints** tab:



General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Tensioner brace and compression bar (13)

Tensioner brace and compression bar (13) creates one or two bracing crosses between two columns or beams. It is possible to add compression bars between the main parts. You can add connections between the main parts and the bracing crosses, and between the main parts and the compression bars.

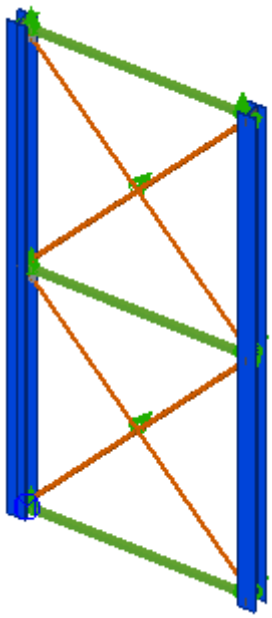
Because you can use other components inside **Tensioner brace and compression bar (13)** to create the connections between parts, the component has a hierarchical component structure. **Tensioner brace and**

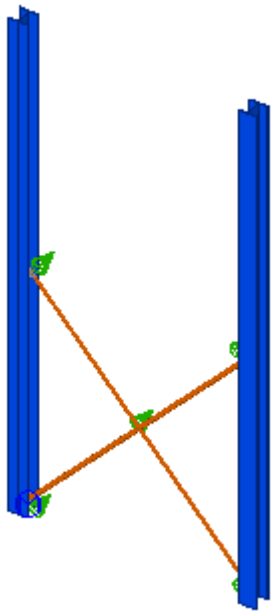
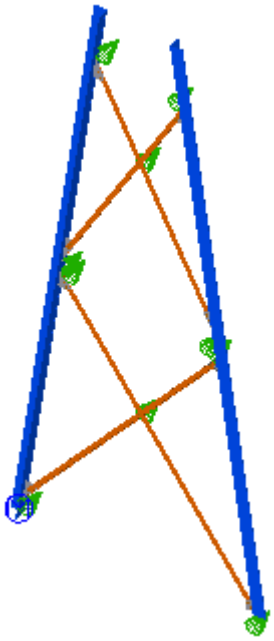
compression bar (13) is on the highest level in the component hierarchy (level 0) and the connections are on a lower level (level 1) in the component hierarchy.

Objects created

- Bracing (1 or 2)
- Compression bars (optional)
- Connections between main parts and bracing
- Connections between main parts and compression bars
- Connections in bracing crosses

Use for

Situation	Description
	Two bracing crosses and three compression bars with connections.

Situation	Description
	One bracing with connections.
	Tapered main parts (mast construction) and two bracing crosses with connections.

Selection order

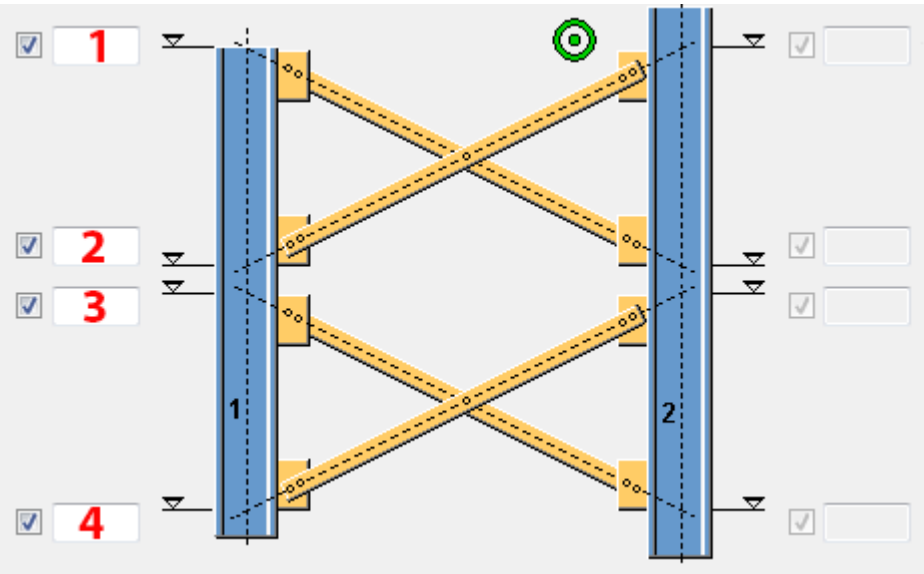
1. Select the first main part (column or beam).
2. Select the second main part (column or beam).

   The component is created automatically when the secondary part is selected.

Picture tab

Use the **Picture** tab to control the bracing levels.

Bracing levels

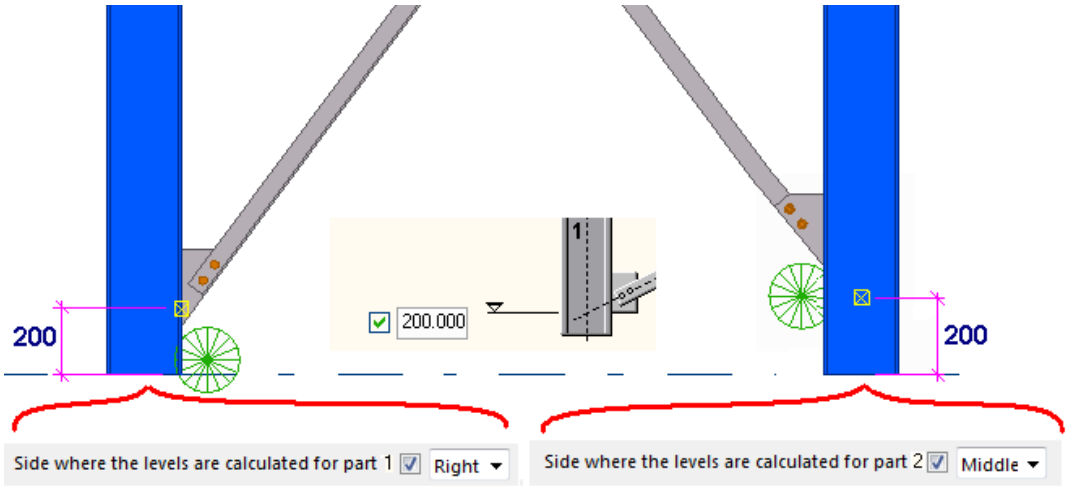


	Description
1	Top level of the upper bracing. If no value is entered, the top bracing is not created.
2	Bottom level of the upper bracing. If no value is entered, the top bracing is not created.
3	Top level of the lower bracing.
4	Bottom level of the lower bracing.

Bracing reference

For both main parts, define the reference of the bracing levels. The reference can be set for both the horizontal and the vertical direction.

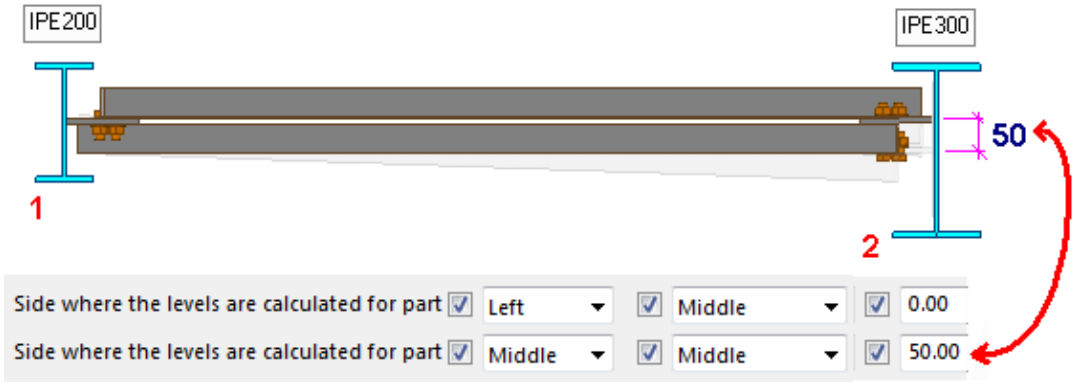
For example:



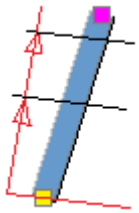
Bracing offset

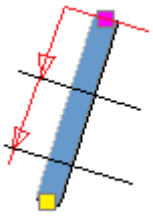
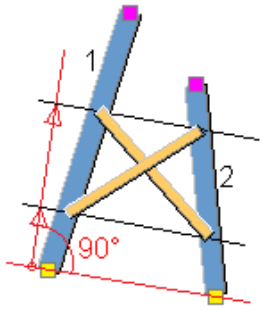
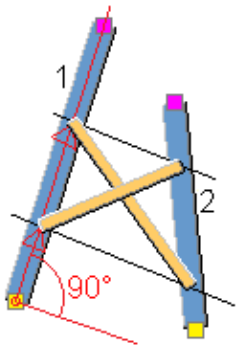
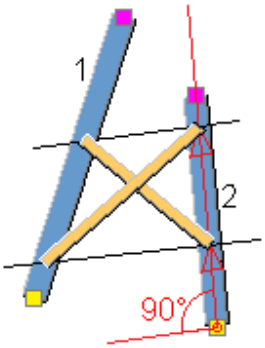
Define the offset perpendicular to the bracing. You can move the created plate or part by entering a value in the x-, y, or z-direction.

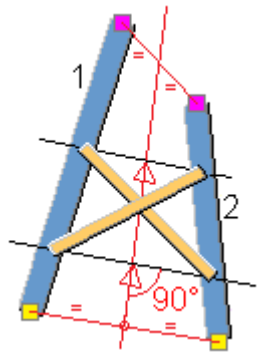
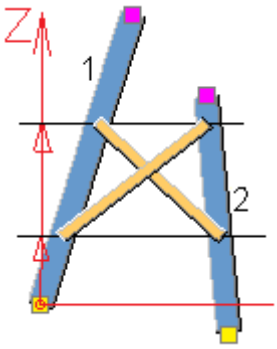
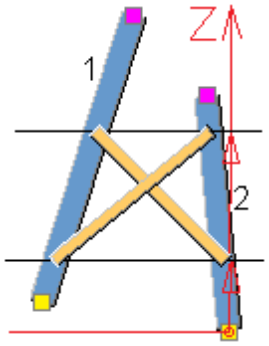
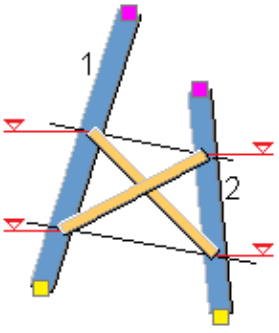
In the example below, the main parts have unequal dimensions and the reference is set to **Middle**. To create the bracing crosses horizontally, an offset must be entered for the biggest part. The needed offset is the height difference between IPE200 and IPE300, divided by 2 = 50mm.



Bracing direction

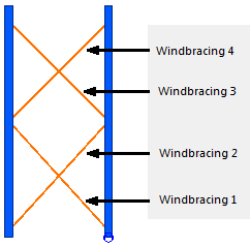
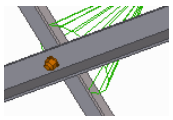
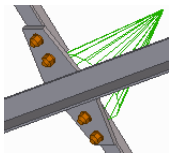
Option	Description		
Direction	Select the direction of the bracing.		Start point of the main part as the reference point.

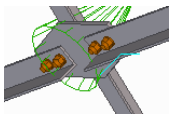
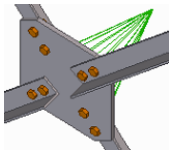
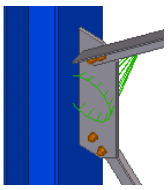
Option	Description		
			End point of the main part as the reference point.
Calculation of levels	Select the reference line used for positioning the bracing when the main parts are not parallel.		Offset from the start point in part 1, perpendicular to the line through the start points.
			Offset from the start point in part 1, in local x-direction.
			Offset from the start point in part 2, in local x-direction.

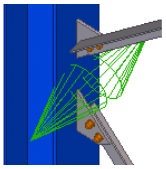
Option	Description		
			Reference line through the start and end points of the main parts, offset from the start of the reference line.
			Offset from the start point in part 1, in z-direction.
			Offset from the start point in part 2, in z-direction.
			Freely defined values.

Parts tab

Use the **Parts** tab to control the properties, bracing position, and rotation. Additionally, you can define bracing splitting and shortening values.

Option	Description
Windbracing 	Define the bracing profile by selecting it from the profile catalog.
Windbracing position in plane	Set the position in plane for the first and the second bracing element.
Windbracing rotation	Select the rotation for the first and the second bracing element. This option is useful when the bracing elements are crossing and they are connected in the crossing.
Windbracing translation	Select the offset of the first and the second bracing element from the reference points. This option is useful when the bracing elements are positioned so that the first bracing is alongside the second bracing. Typically, the first bracing element is set to **Forwards** and the second bracing element to **Backwards**.
Splitting the diagonal bracing	Select whether the diagonal bracing elements are split or connected with a component. Define the connecting component on the **Joints** tab by typing the number of the component in the **Connect diagonals with joint number** box. Bracing is not split.  Possible component for connecting the diagonal bracing: **Seating (30)**. 

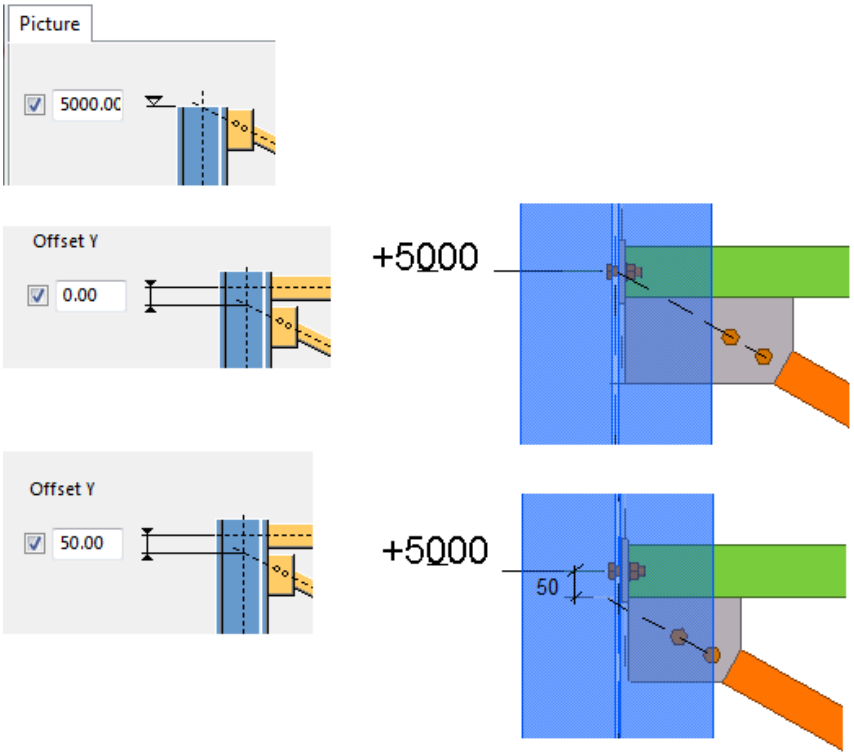
Option	Description		
		First diagonal bracing element is split. Possible component for connecting the diagonal bracing: **Bolted gusset (11)**.	
		 Second diagonal bracing element is split. Possible component for connecting the diagonal bracing: **Bolted gusset (11)**.	
		 Both diagonal bracing elements are split. Possible component for connecting the diagonal bracing: **Central gusset (169)**.	
Connecting bracing crosses	Select whether the gusset plates of two bracing crosses above each other are connected. Define the connecting component on the **Joints** tab by typing the number of the component in the **Connection number** box.	 Bracing crosses are connected with a gusset connection. Possible component for connecting the diagonal bracing: **Bolted gusset (11)**.	

Option	Description
	 Bracing crosses are not connected. Separate connection is created for each diagonal bracing. Possible component for connecting the diagonal bracing: **Bolted gusset (11)**. 
Distance between windbracings	Define the distance between bracing elements. If the bracing elements are crossing each other, this value typically defines the gusset plate thickness.
Shorten windbracings	Define how much the bracing are shortened. The entered value is written in the user-defined attributes of the bracing. The value is used in drawings.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Compression bar tab

Use the **Compression bar** tab to define the compression bar properties.

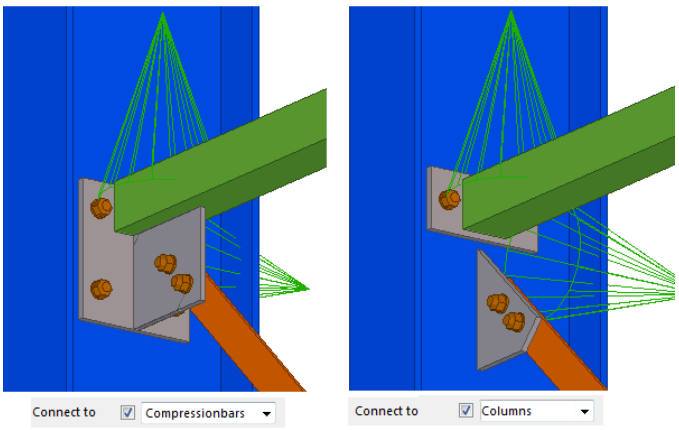
Option	Description
Compression bar	Define the compression bar thickness, width and height.
Create bar	Select whether the compression bar is created. You can define up to three compression bars.
Y offsets **X offsets**	Define the horizontal and vertical offset of the bracing. Reference is the level defined on the **Picture** tab. For example: 
Position in plane **Rotation** **Position in depth**	Select the orientation of the compression bars.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Joints tab

Use the **Joints** tab to define the components used for connecting the main parts and the diagonal bracing elements and the compression bars.

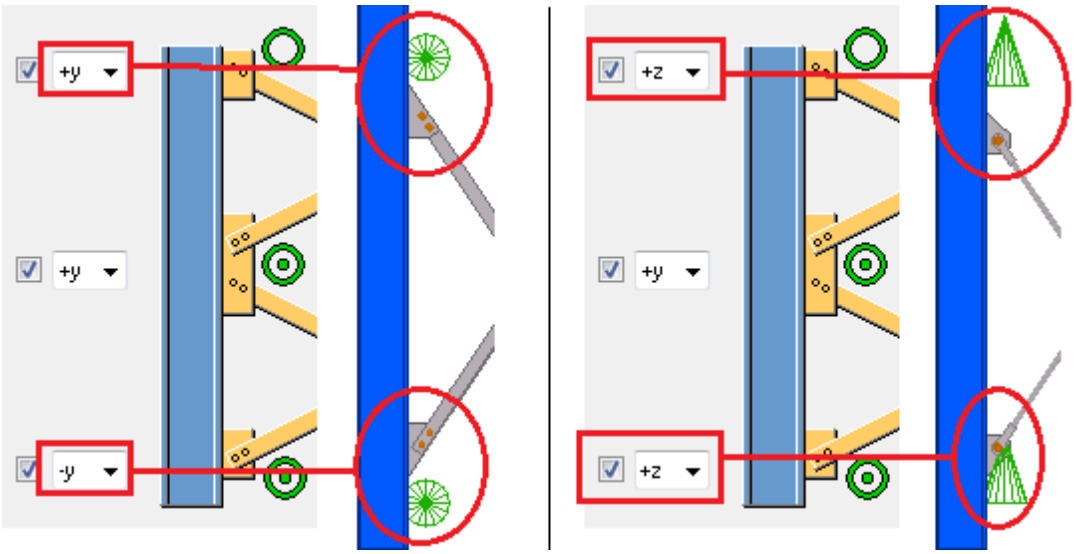
Option	Description
Creation of joints	Select whether connections are created between the bracing elements. **No**: Only the bracing elements are created. **Yes**: Components are added between the bracing elements.
Connect to	Select whether the bracing elements are connected to the main parts or the compression bars. For example:

Option	Description
	
Connection number **Connect diagonals with joint number** **Userjoint application number** **Joint direction**	Type the component number that is used for connecting the bracing, the diagonal bracing cross, or the compression bar, and the component application number. Select the connection direction for the diagonal bracing cross. • Default component for the bracing is **Bolted gusset (11)**. • Default component for the diagonal bracing cross is **Seating (30)**. • Default component for the compression bar is **End plate (144)**.
Configuration file for the joint	Configuration setting for the connection. For example, if you type `CS_M13`, it means that a setting named `CS_M13` must be available for the used connection.
Tensioner position	Define the tensioner position if the **Tensioner (7)** connection is used.

Joints dir tab

Use the **Joints dir** tab to control the up directions of the connections used between the main parts and the diagonal bracing, and the up direction of the connections used between the main parts and the compression bars.

In the example below, **Seating (30)** has been defined as the connecting component on the **Joints** tab:



UDA tab

Use the **UDA** tab to add information in the user-defined attributes (UDAs) of the parts.

Option	Description
Part	Select to which part the related information can be saved.
UDA name	Enter the name of the user-defined attribute.
Type	Select the UDA type. Use **String** for text, **Integer** for numbers, **Float** for numbers with decimals and **Option** for selecting an item in a list.
Value	Enter the value that is saved to the user-defined attribute. Use text and/or numbers, depending on the defined UDA type.

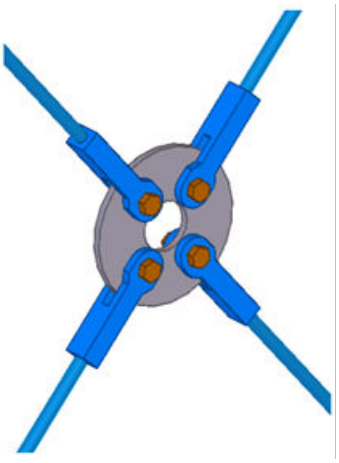
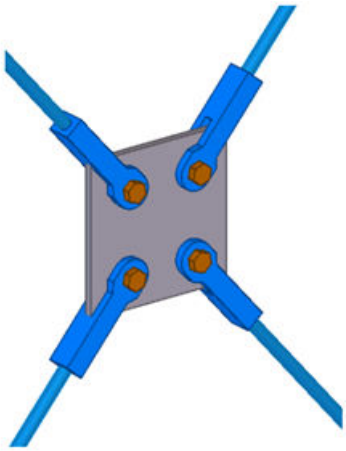
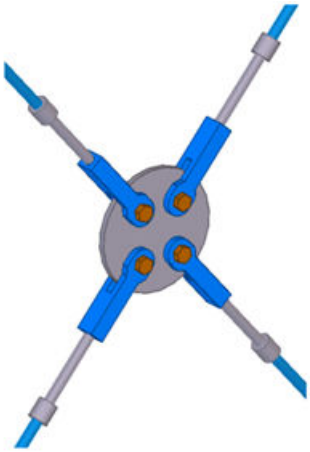
Tensioner central gusset (18)

Tensioner central gusset (18) creates a gusset plate to connect bracing bars.

Objects created

- Gusset plate
- Fork or plate
- Tensioners (optional)
- Bolts
- Welds

Use for

Situation	Description
	Round gusset plate with a hole.
	Square gusset plate.
	Round gusset plate. Tensioners are created.

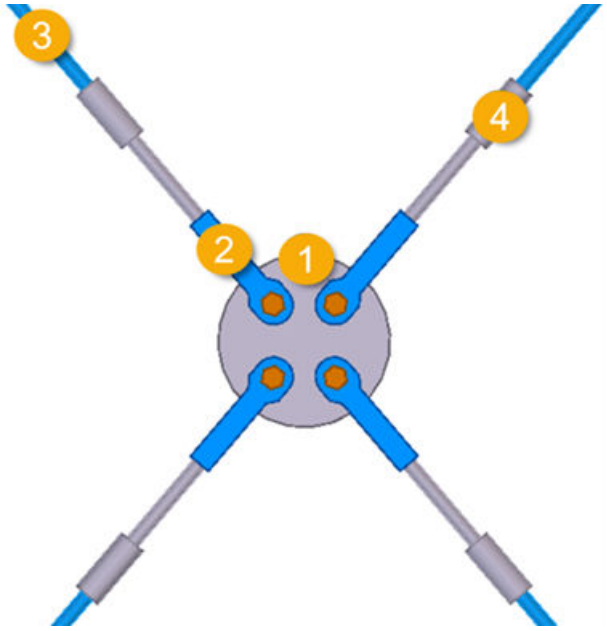
Before you start

Model at least 3 bracing bars.

Selection order

1. Select the bracing bars.
2. Click the middle mouse button to create the central gusset plate and the forks.

Part identification key

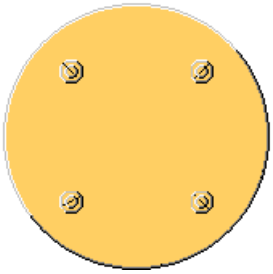
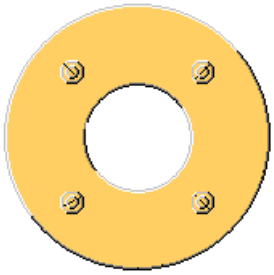
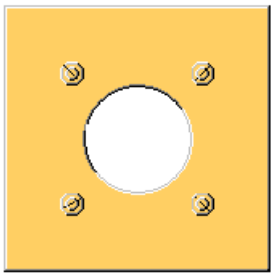
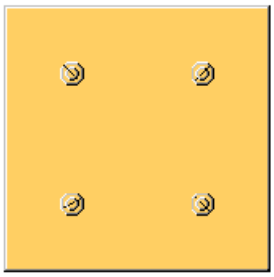


	Description
1	Gusset plate
2	Fork
3	Bracing bar
4	Tensioner

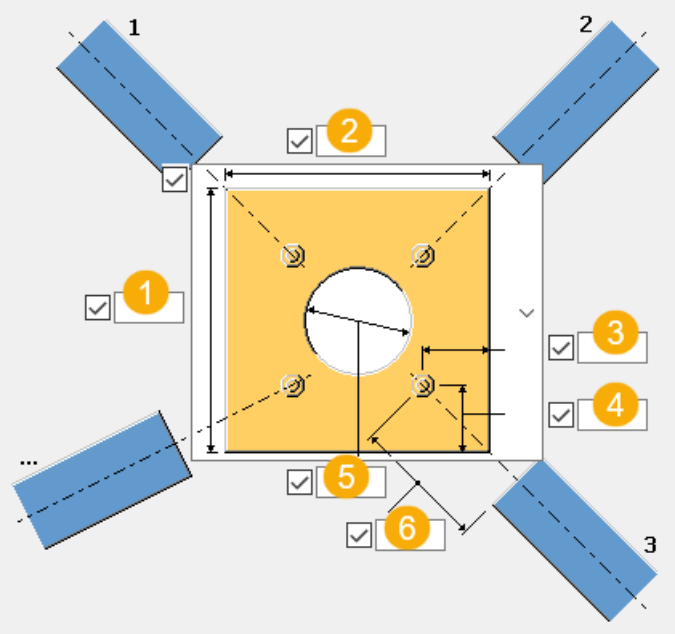
Picture tab

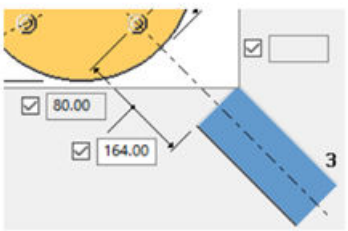
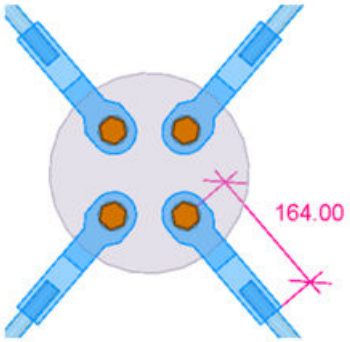
Use the **Picture** tab to define the shape and the dimensions of the central plate.

Gusset plate shape

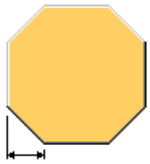
Option	Description
	Circular gusset plate
	Circular gusset plate with a hole
	Square gusset plate with a hole
	Square gusset plate

Gusset plate dimensions



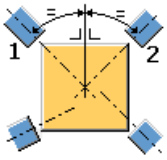
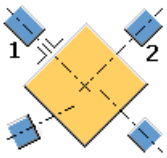
	Description
1	Height of the gusset plate.
2	Width of the square gusset plate.
3	Bolt edge distance. Horizontal bolt edge distance for square gusset plates.
4	Vertical bolt edge distance for square gusset plates.
5	Diameter of the hole in the gusset plate.
6	Fork length, calculated from the center of the hole to the end of the fork.  

Chamfer shape and dimension

Option	Description
	No chamfers
	Line chamfer Define the chamfer dimension.
	Convex chamfer Define the chamfer dimension.

Gusset plate position

Select the position of the gusset plate relative to the bracing bars.

Option	Descriptioin
	Middle of the corner between the first and the second bar.
	Perpendicular to the first bar.

Create plate as

- **Contour plate**. The profile can be `PL12`, for example.
- **Beam**. The profile can be `PL300*300`, for example.

  Enter the prefix of the beam profile. You can only enter the prefix if the plate is created as a beam profile.

Fork tab

Use the **Fork** tab to define the fork properties.

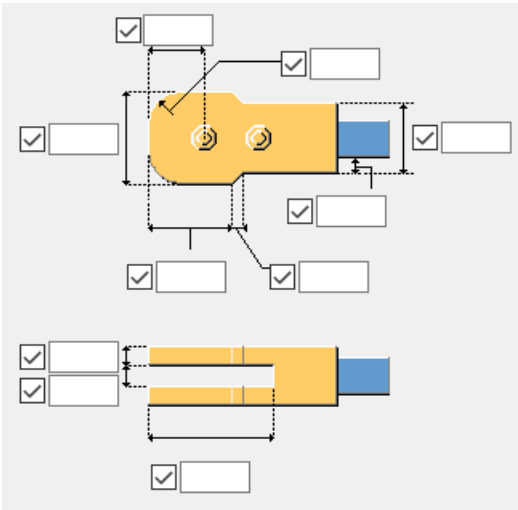
Parts

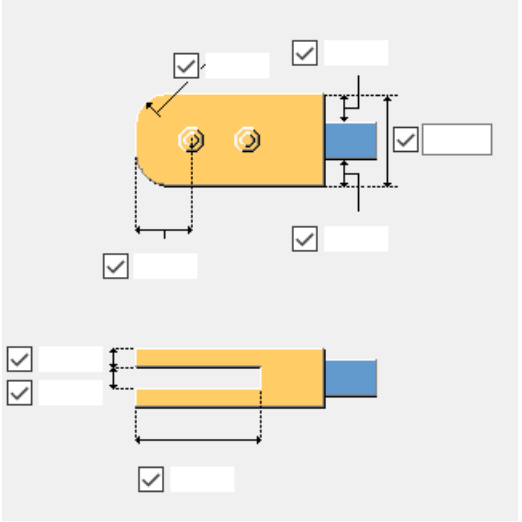
Option	Description
Plate	Thickness of the fork.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Part shape and dimensions

Select the shape of the fork: **Part 1** or **Part 2**.

Options	Description
	Define the dimensions of the reduced fork.

Options	Description
	Define the dimensions of the fork.

Parameters tab

Use the **Parameters** tab to define the end plate properties, shape, and dimensions.

Parts

Option	Description
End plate	Thickness, width, and height of the end plate.

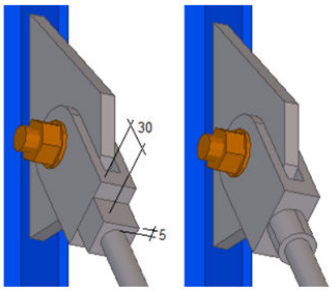
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Option	Description	Default
Finish	Describes how the part surface has been treated.	

End plate shape

Option	Description
	Default Square AutoDefaults can change this option.
	Square
	Round

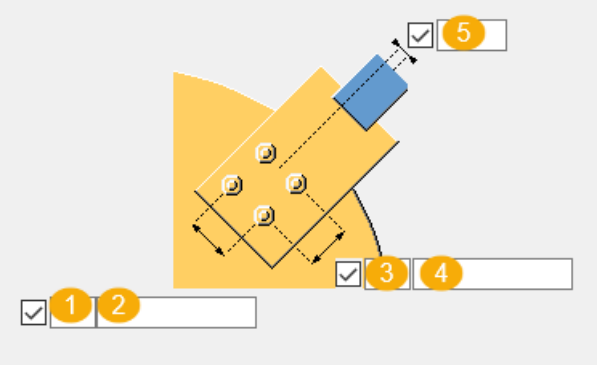
End plate offset

Option	Description	Example
	Define the end plate offset from the bracing bar.	End plate thickness is set to `30` and the offset it set to `5`. 

Bolts tab

Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions



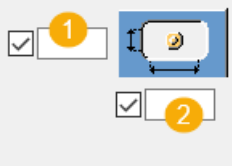
	Description
1	Number of bolts in the longitudinal direction (axial to the bar).
2	Bolt spacing in the longitudinal direction.
3	Number of bolts in the transverse direction.
4	Bolt spacing in the transverse direction.
5	Bolt group offset in the transverse direction.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	`20 mm`
Bolt standard	Bolt standard to be used inside the component.	`4014-8.8`
Tolerance	Gap between the bolt and the hole.	`3 mm`
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Assembly type	Location where the bolts should be attached.	`Site`

Slotted holes

You can define the dimensions of the slotted holes in the horizontal and vertical direction, and a gap for oversized R tapped holes. The default is `0` mm, which creates a circular hole.

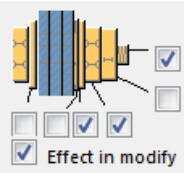


	Description
1	Vertical dimension of the slotted hole.
2	Horizontal dimension of the slotted hole.

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Tensioner T tab

Use the **Tensioner T** tab to define the properties and dimensions of the tensioners.

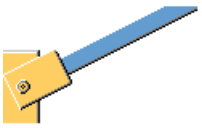
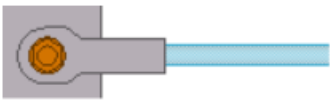
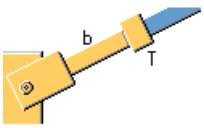
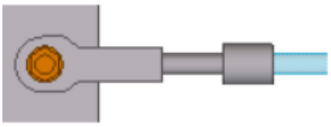
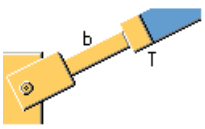
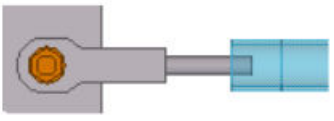
Parts

Option	Description	Default
Tensioner T	Select the profile from the profile catalog.	D40

Option	Description	Default
Part B	Select the profile from the profile catalog. This is the bar profile between the fork and the tensioner.	

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Create tensioner

Opion	Description	Example
	Tensioner is not created.	
	Tensioner is created.	
	Tensioner is created. The stopper part is added to the bracing.	

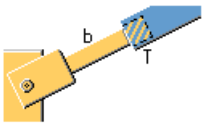
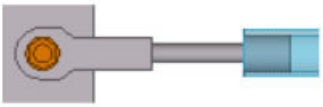
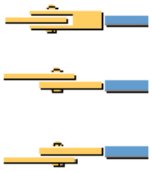
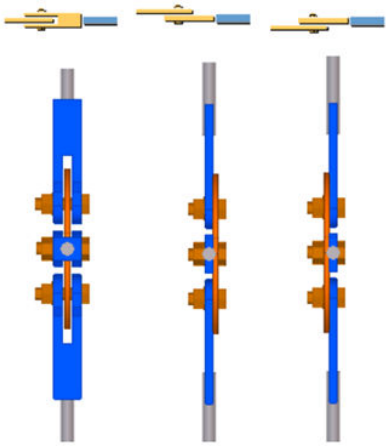
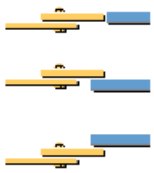
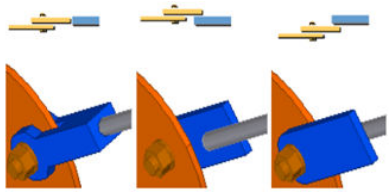
Opion	Description	Example
	Used for compression tubes.	
	Tensioner is created. The stopper part is placed inside the compression tube.	

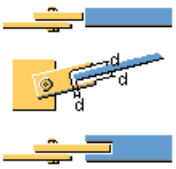
Plate position

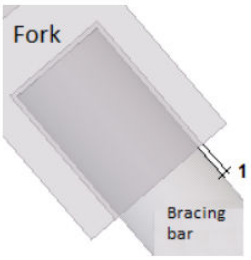
Option	Descriptino	Example
	Define the position of the plate on the fork.	

Fork position

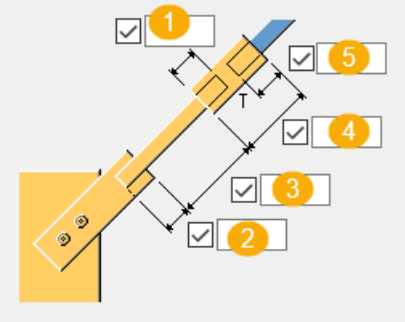
Option	Description	Example
	Define the position of the fork on the bracing bar.	

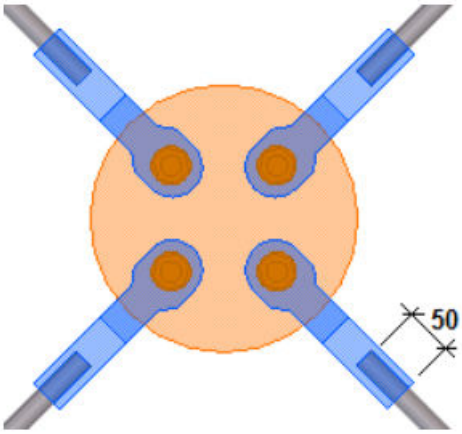
Opening in fork

Option	Description	Example
	Select whether an opening is created in the fork. The opening is always square. You can define the opening if both the tensioner and an extra	

Option	Description	Example
	bracing bar are created.	
	Define the gap for the opening.	The default value is 1 mm. 

Dimensions

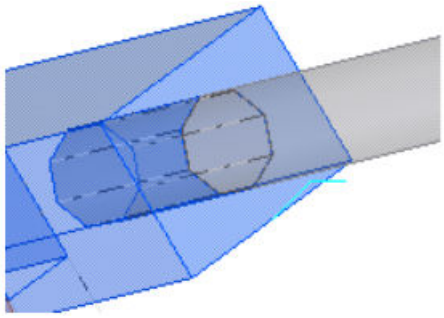
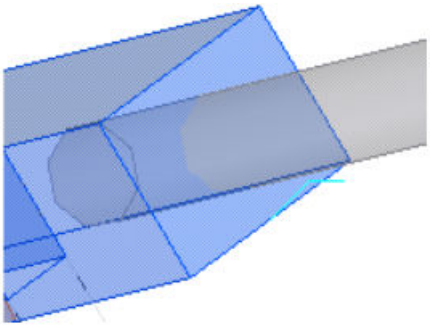


	Description	Default
1	Define an overlap for the extra bracing bar in the tensioner.	0 mm
2	Define an overlap for the bracing bar. 	
3	Define the length of the extra bracing bar between the fork and the tensioner.	300 mm

	Description	Default
4	Define the length of the tensioner.	40 mm
5	Define an overlap for the bracing bar in the tensioner.	0 mm

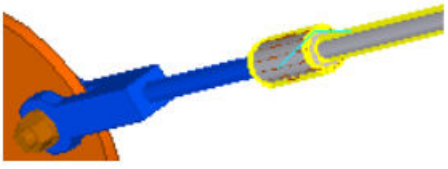
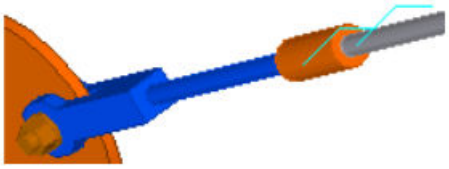
Cut part B in fork

Select whether the fork is cut if the bracing bar goes through the fork. The fork cut adapts to the size of the bracing bar.

Example	Description
	Part B is cut.
	Part B is not cut.

Tensioner T add to secondary

Select whether the tensioner is added to the bracing bar, or handled as a loose part and welded to the bracing bar.

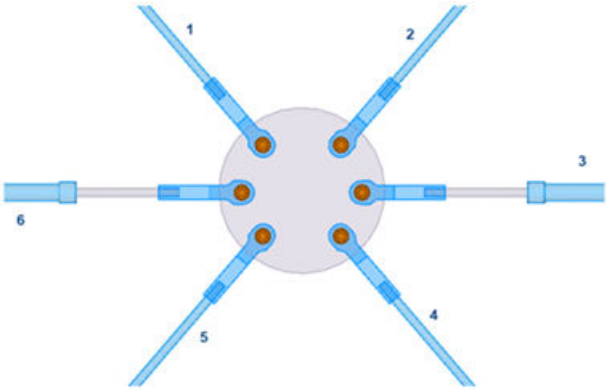
	Select **Yes** to add the tensioner to the bracing bar.
	Select **No** to weld the tensioner to the bracing bar.

Extra tensioners

Use the **Extra tensioners** tab to define the properties and dimensions of extra tensioners. You can define two types of tensioners. You can define extra tensioners if more than one bracing bar is connected. If there is only one bracing bar, the settings on the **Tensioner T** tab are used for tensioners.

Parts

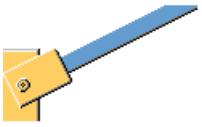
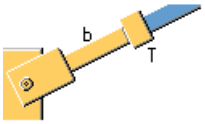
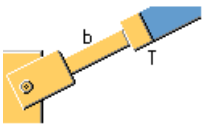
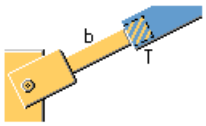
Option	Description
Tensioner T	Select the profile from the profile catalog.
Part B	Select the profile from the profile catalog. This is the bar profile between the fork and the tensioner.

Option	Description
Apply on tensioner number	Define the bars on which the setting for the extra tensioners is applied. Use a space to define several bar numbers. 

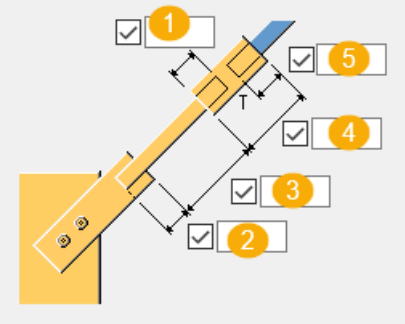
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the

Option	Description	Default
		Components settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Create tensioner

Option	Description
	Tensioner is not created.
	Tensioner is created.
	Tensioner is created. The stopper part is added to the bracing. Used for compression tubes.
	Tensioner is created. The stopper part is placed inside the compression tube.

Dimensions



	Description
1	Define an overlap for the extra bracing bar in the tensioner.
2	Define an overlap for the bracing bar.

	Description
3	Define the length of the extra bracing bar between the fork and the tensioner.
4	Define the length of the tensioner.
5	Define an overlap for the bracing bar in the tensioner.

Cut part B in fork

Select whether the fork is cut if the bracing bar goes through the fork (**Yes**). The fork cut adapts to the size of the bracing bar.

Tensioner T add to secondary

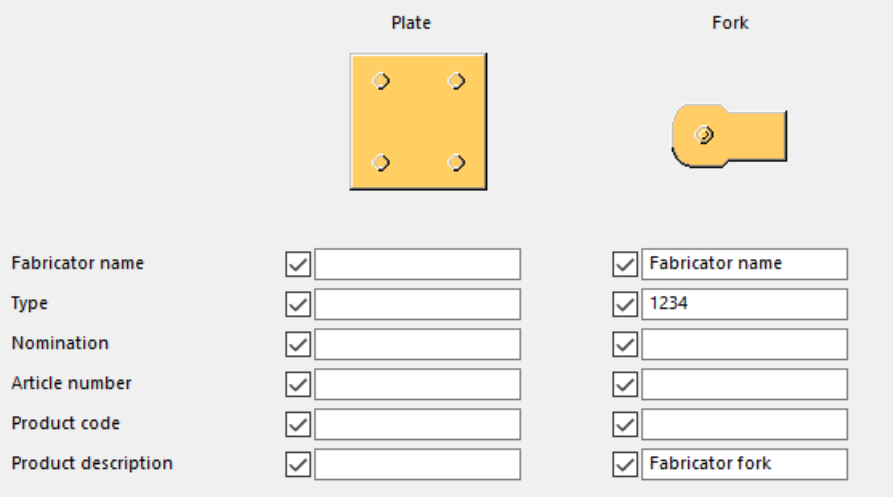
Select whether the tensioner is added to the bracing bar (**Yes**), or handled as a loose part and welded to the bracing bar.

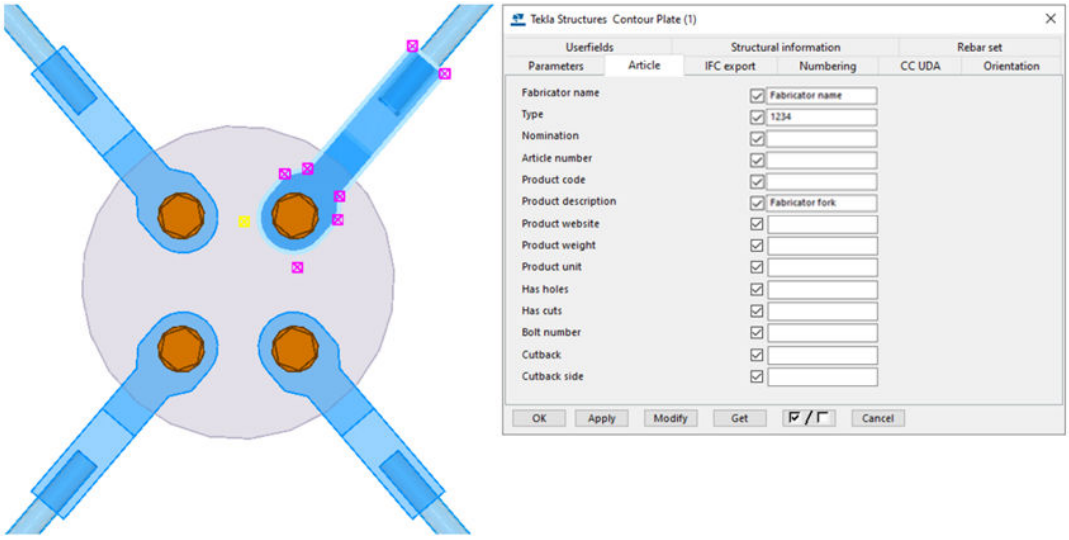
UDA tab

Use the **UDA** tab to define user-defined attributes (UDA). The attributes are written to the central gusset and forks.

Example

Define the needed user-defined attributes.





General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

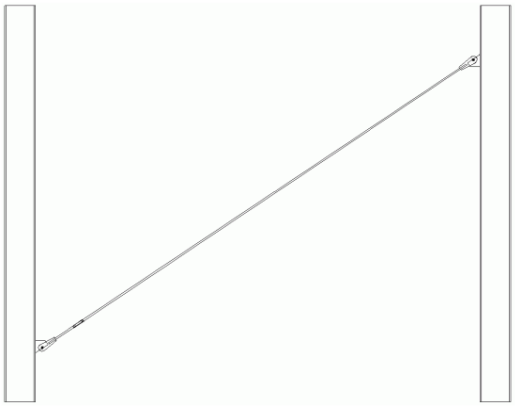
Turnbuckle bracing (S3)

Turnbuckle bracing (S3) creates a turnbuckle assembly, rods, and end connections.

Objects created

- Turnbuckle
- Rod
- Fastener plate
- Gusset plate
- Bolts

Use for

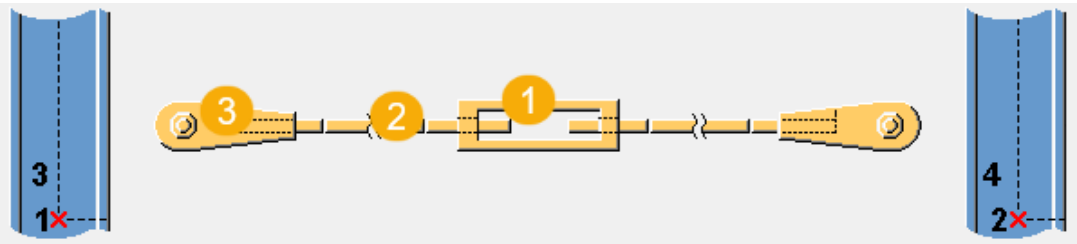
Situation	Description
	Turnbuckle connection

Selection order

1. Pick the first point.
2. Pick the second point.
3. Select the first part.
4. Select the second part.

   The connection is created automatically when the second part is selected.

Part identification key

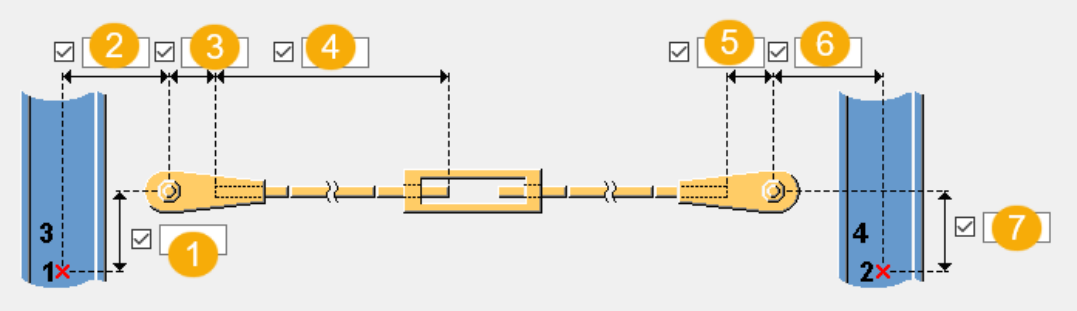


	Description
1	Turnbuckle
2	Rod
3	Fastener plate/Gusset plate

Picture tab

Use the **Picture** tab to define the turnbuckle dimensions.

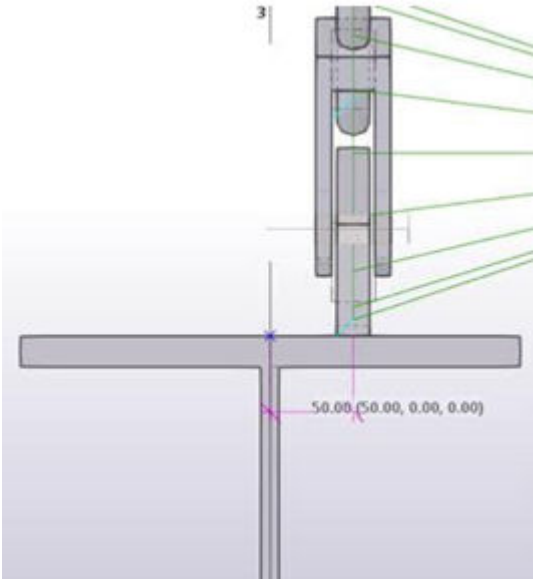
Dimensions



	Description	Default
1	Offset between the first picked position and the actual generation point.	0 mm
2	Distance from the first bolt to the center line of the first part.	First part collision point + 100 mm
3	Distance from the first bolt to the end of the first rod. Used with battledores or clevises.	100 mm
4	Fixed length of the rod. The length of the second rod is automatically calculated.	400 mm
5	Distance from the first bolt to the end of the second rod. Used with battledores or clevises.	100 mm
6	Distance from the first bolt to the center line of the second part.	Second part collision point + 100 mm
7	Offset between the first picked position and the actual generation point.	0 mm

Rod depth offset

Define the fastener/gusset plate offset from the part center line. By default, the fastener plate/gusset plate is located in line with the part center line.



Turnbuckle tab

Use the **Turnbuckle** tab to define the turnbuckle dimensions, thread type and cuts, and whether the turnbuckle is created as an assembly or as single parts.

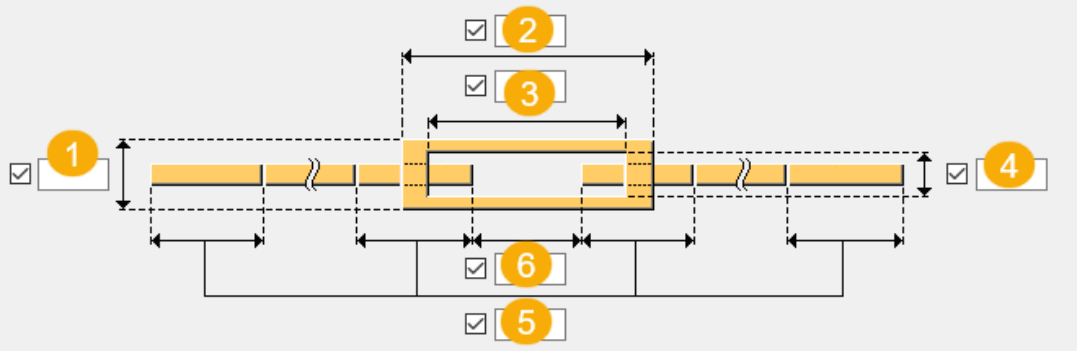
Parts

Option	Description
Rod	Select the rod profile from the profile catalog.
Turnbuckle	Select whether to create a turnbuckle.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the

Option	Description	Default
		Components settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Dimensions



	Description	Default
1	Outer diameter of the turnbuckle	Diameter of the rod + 2 x turnbuckle thickness
2	Length of the turnbuckle	`50 mm`
3	Length of the turnbuckle opening	Length of the turnbuckle – 20 mm
4	Width of the turnbuckle opening	Rod thickness – 1
5	Length of the thread cut	`100 mm`
6	Gap between the rods	`100 mm`

Thread type

Define the direction of the threads at the turnbuckle.

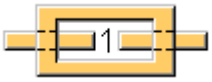
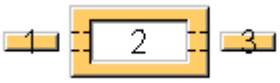
Option	Description
	RH – LH (right hand – left hand) Tekla Structures stores RH as a user-defined attribute to the first rod and LH as a user-defined attribute to the second rod.
	LH – RH (left hand – right hand) Tekla Structures stores LH as a user-defined attribute to the first rod and

Option	Description
	RH as a user-defined attribute to the second rod.

Create thread cuts

Option	Description
	Threads are not cut.
	Threads are cut.

Assembly / single part turnbuckle

Option	Description
	Assembly
	Single parts

Connection tab

Use the **Connection** tab to define the plate properties and the connection type.

Parts

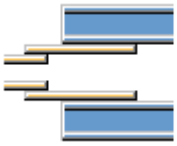
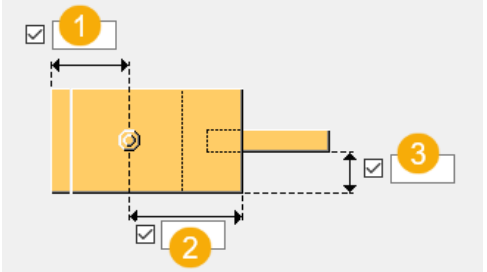
Option	Description
Fastener plate	Thickness, width and height of the fastener plate.
Gusset plate	Thickness, width and height of the gusset plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part**

Option	Description	Default
		material box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Connection type

Select the connection type: **Battledore**, **Clevis**, **Bolted gusset**, or **Tube gusset**.

Option	Description	Default
Battledore side	Select whether the battledore and gusset are generated in front of or behind the rod generation plane. 	Front
Battledore dimensions	 **1** Distance from the bolt to the edge of the fastener plate. **2** Distance from the bolt to the gusset plate edge. **3** Clearance between the gusset corner and the rod.	**1** 60 mm **2** 2.5 x bolt diameter **3** (Fastener plate width – Rod diameter) / 2
Clevis pin size	Clevis pin size is the value stored as a free attribute to the clevis. Used only with clevis.	Bolt diameter + 4 mm
Connection setting	User-defined attribute file for the bolted gusset or tube gusset connection.	

Bolts tab

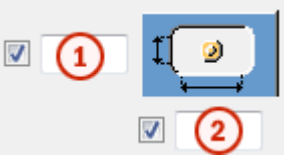
Use the **Bolts** tab to define the bolt properties.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.



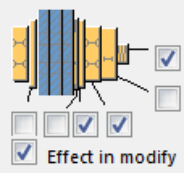
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	

Option	Description	Default
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

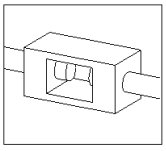
Muffe (26)

Muffe (26) connects two beams with a rectangular plate that has a hole in the middle.

Objects created

- Rectangular plate (muffe)

Use for

Situation	Description
	Two beams connected with a rectangular plate that has a hole in the middle.

Selection order

1. Select the main part.
2. Select the secondary part.

   The connection is created automatically when the secondary part is selected.

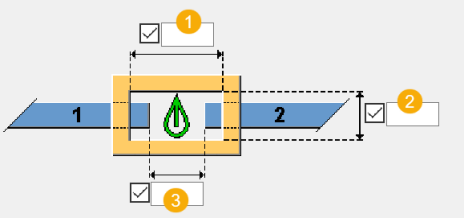
Part identification key

	Description
1	Rectangular plate with a hole

Picture tab

Use the **Picture** tab to define the connection dimensions.

Dimensions



	Description	Default
1	Horizontal dimension of the hole.	`0.5 * muffe height`
2	Vertical dimension of the hole.	`Muffe thickness - 30 mm`
3	Distance between the main part and the secondary part.	`0.5 * muffe height - 50 mm`

Parts tab

Use the **Parts** tab to define the plate properties.

Muffe

You can use a profile or a custom component from the **Applications & components** catalog and define the custom settings, up direction, and rotation.

Option	Description	Default
Muffe	Thickness, width, and height of the plate.	Thickness and width: `100 mm` Height: `200 mm`

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

Use the **Parameters** tab to define the bolt hole tolerance.

Tolerance

Option	Description
Tolerance of bolt hole	Define the tolerance value. By default, the tolerance is zero.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Design tab

Click the link below to find out more:

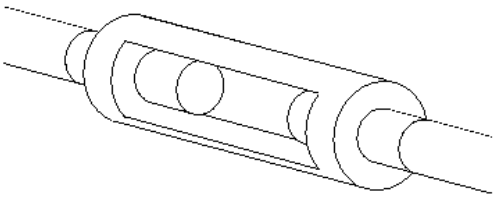
Turnbuckle connection (126)

Turnbuckle connection (126) creates a turnbuckle assembly. The parts that are connected must be parallel. Tekla Structures creates the connection objects symmetrically.

Objects created

- Turnbuckle

Use for

Situation	Description
	Turnbuckle connection

Selection order

1. Select the main part.
2. Select the secondary part.

   Note that the parts must be parallel.

   The connection is created automatically when the secondary part is selected.

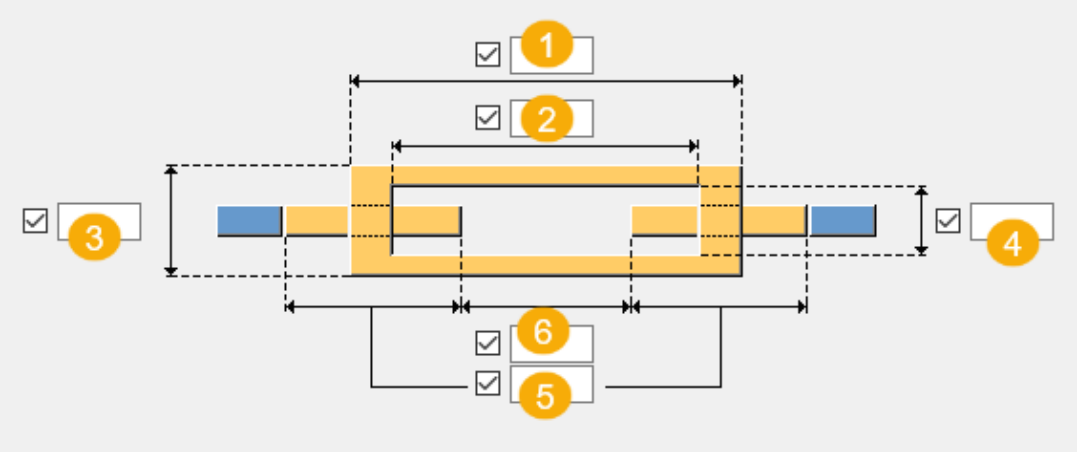
Part identification key

	Description
1	Turnbuckle

Picture tab

Use the **Picture** tab to define the dimensions of the turnbuckle.

Dimensions



	Description	Default
1	Length of the turnbuckle	`50 mm`
2	Length of the turnbuckle opening	Length of the turnbuckle – 20 mm
3	Outer diameter of the turnbuckle	Diameter of the rod + 2 x turnbuckle thickness
4	Width of the turnbuckle opening	Rod thickness - 1
5	Length of the thread cut	`100 mm`
6	Distance between the thread cuts	Half of the turnbuckle length

Parts tab

Use the **Parts** tab to define whether the turnbuckle is created as a profile or as a custom component.

Create as

Use this option to select whether to create the turnbuckle as a profile or as a custom component.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

If you select to create the turnbuckle as a custom component, select the custom component from the **Applications & components** catalog and define the custom settings, up direction, rotation, and length.

Parameters tab

Use the **Parameters** tab to define the thread type, cuts and size, and whether the turnbuckle is created as an assembly or as single parts.

Thread type

Define the direction of the threads at the turnbuckle.

Option	Description
	Default RH – LH (right hand – left hand) Tekla Structures stores RH as a user-defined attribute to the first rod and LH as a user-defined attribute to the second rod. AutoDefaults can change this option.
	RH – LH (right hand – left hand) Tekla Structures stores RH as a user-defined attribute to the first rod and LH as a user-defined attribute to the second rod.

Option	Description
	LH – RH (left hand – right hand) Tekla Structures stores LH as a user-defined attribute to the first rod and RH as a user-defined attribute to the second rod.

Create thread cuts

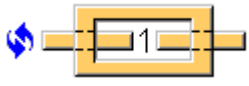
Option	Description
	Default Threads are cut. AutoDefaults can change this option.
	Threads are cut.
	Threads are not cut.

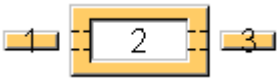
Thread size

When you have selected that threads are not cut, you can select the size of the thread and create the thread 1 mm smaller.

Option	Description
	Default Thread size is not changed. AutoDefaults can change this option.
	Thread size is not changed.
	Thread is created 1 mm smaller.

Assembly / single part turnbuckle

Option	Description
	Default Assembly AutoDefaults can change this option.
	Assembly

Option	Description
	Single parts

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

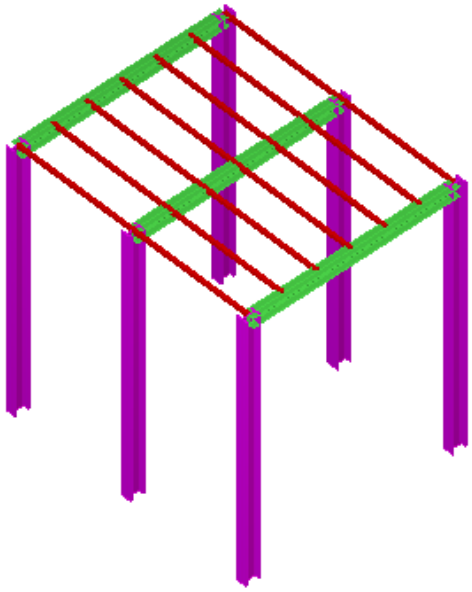
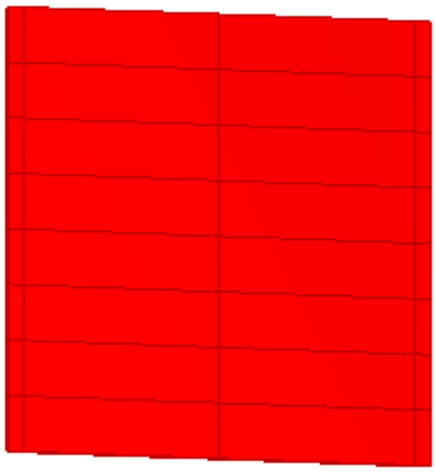
Generation of purlins (50)

Generation of purlins (50) creates multiple profiles that can be used as wall or roof purlins, panels, or timber or concrete parts.

Objects created

- Purlins

Use for

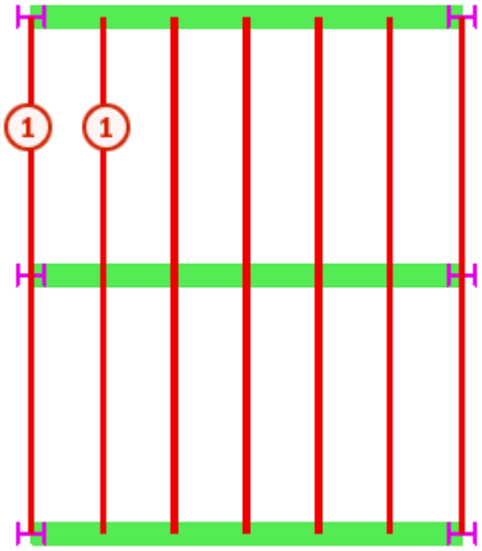
Situation	Description
	Purlins
	Wall panel

Selection order

1. Pick the start point of the purlins.
2. Select the parts that divide up the purlins.
3. Click the middle mouse button to create the purlins.

NOTE The placement of the purlins is defined by the input parts.

Part identification key

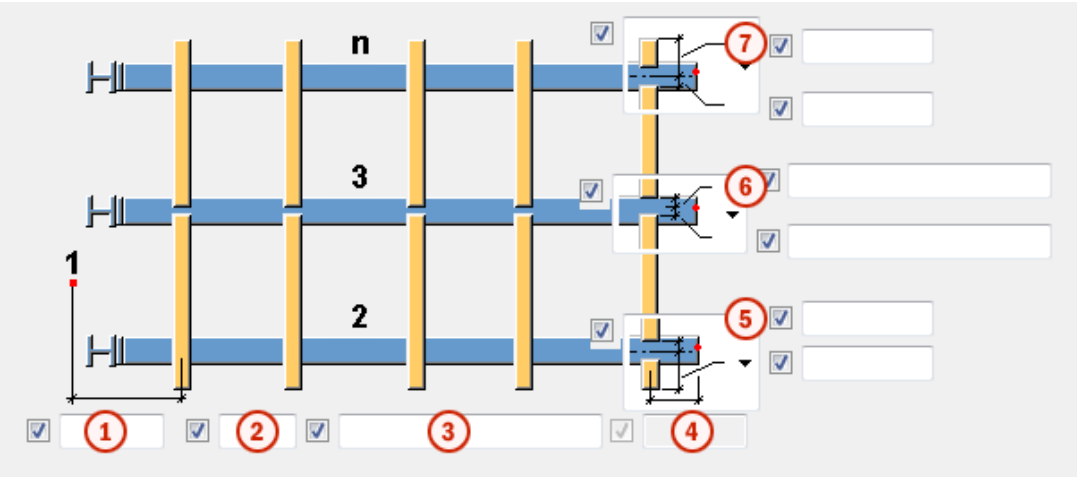


	Part
1	Purlin

Picture tab

Use the **Picture** tab to control the number of purlins, distances between the purlins and purlin overhang lengths.

Purlin dimensions

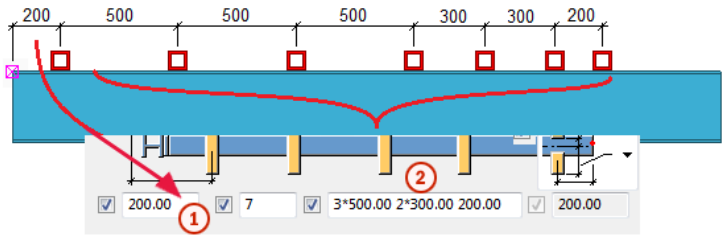
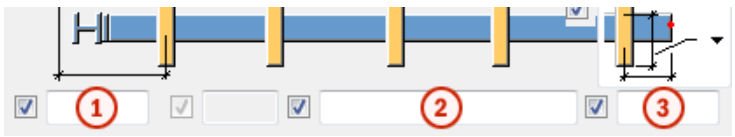


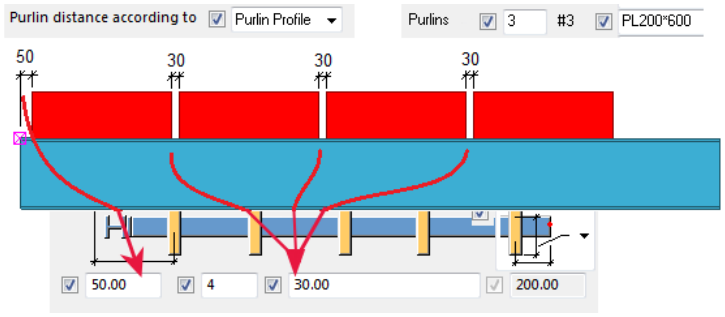
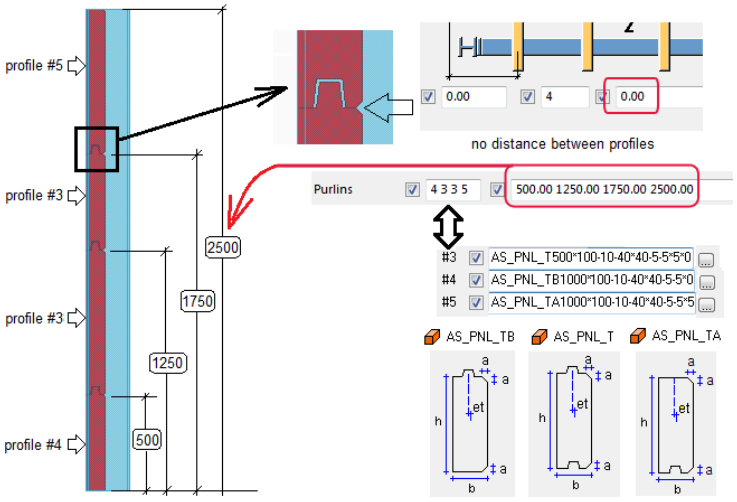
	Description
1	Define the edge distance from the picked point to the first purlin.

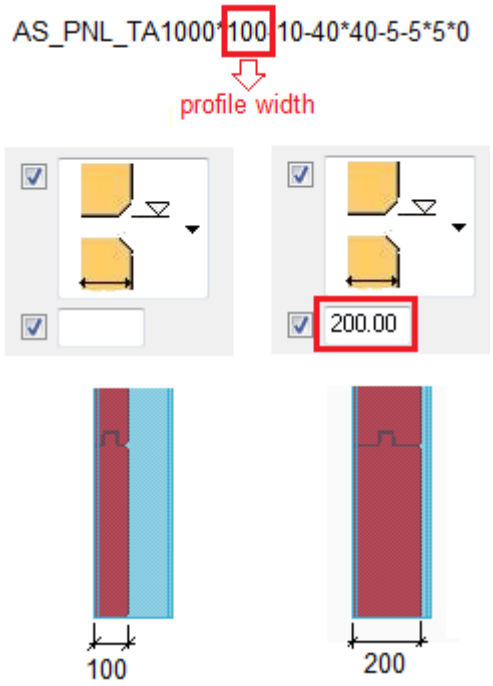
	Description
2	Define the number of purlins.
3	Define the distance between the purlins. The distances are calculated from center-to-center.
4	Define the edge distance from the last purlin to the part end point. The distance depends on the option **Purlin distance according to**.
5	Define the type and length for purlin overhangs in the purlin start.
6	Define how the purlins are split and the size of the gap between the purlins.
7	Define the type and length for purlin overhangs in the purlin end.

Purlin distances

Define how the distances between purlins are calculated.

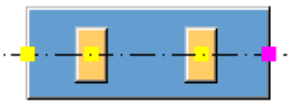
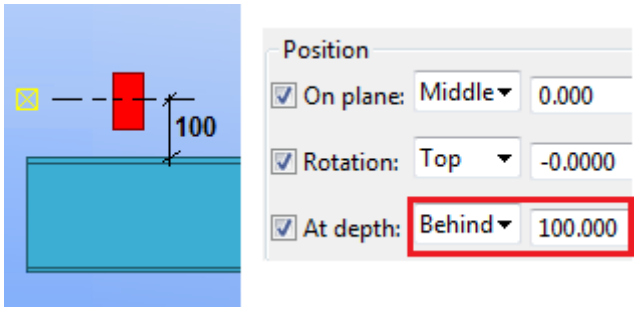
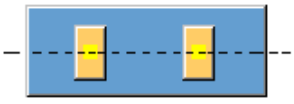
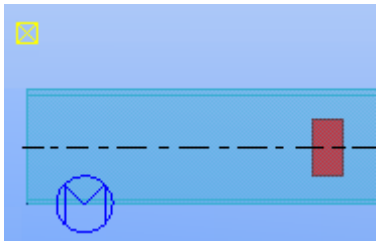
Option	Description
Exact number of purlins	Define the purlin start point with an edge distance ①. The remaining length is divided according to the values in ②. For example: 
Fill to the end	Define the purlin start point with an edge distance in ①. The remaining length is divided according to the values in ②, considering the edge distance in ③. 

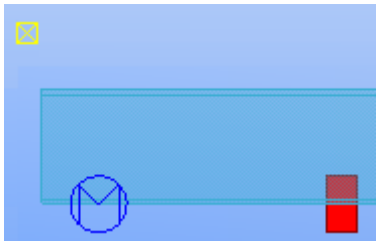
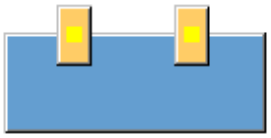
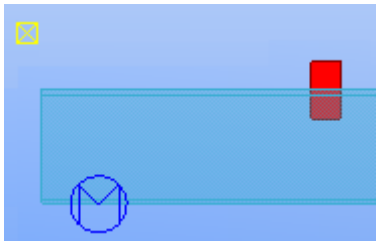
Option	Description
Purlin profile	Intermediate distances Define the distances between the purlins. This is suitable especially for concrete floors. For example, define the distances on the **Picture** tab and the purlin profile properties on the **Parts** tab: 
Divide equally	Purlin distances are equally divided by the length of profiles.
Purlin type	This option is recommended for vertical profiles, such as concrete walls or panels. The height of the part that is set in the purlin profile properties on the **Parts** tab can be overridden by the levels defined in the **Top level purlin panels** option on the **Parts** tab. For example:  The width of the panel that is set in the purlin profile properties on the **Parts tab** can be overridden by the **Plate thickness** option on the **Parts tab**. If the **Plate thickness** option is empty, then the plate thickness in the profile properties is used.

Option	Description
	For example: 

Level

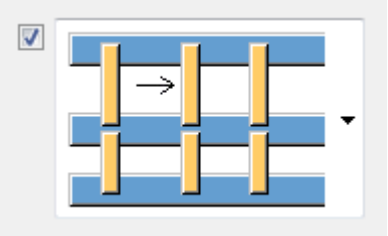
Define the purlin level compared to the main part.

Option	Description
	Reference point of main part 
	Centerline of main part 

Option	Description
	Bottom face of main part 
	Top face of main part 

Purlin direction

Define which direction is used for the new purlins.



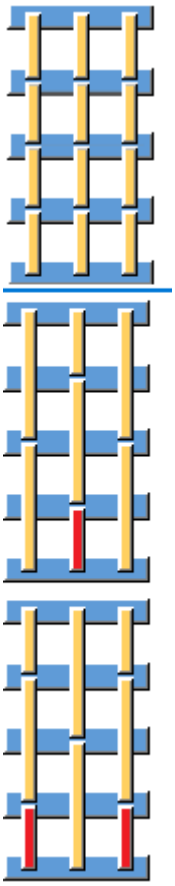
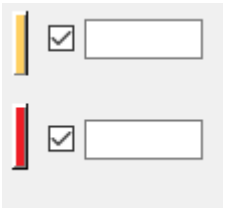
Purlin overlap

Select whether purlins can overlap (**Yes**) or not (**No**). Setting purlins to overlap helps in root sheeting, for example.

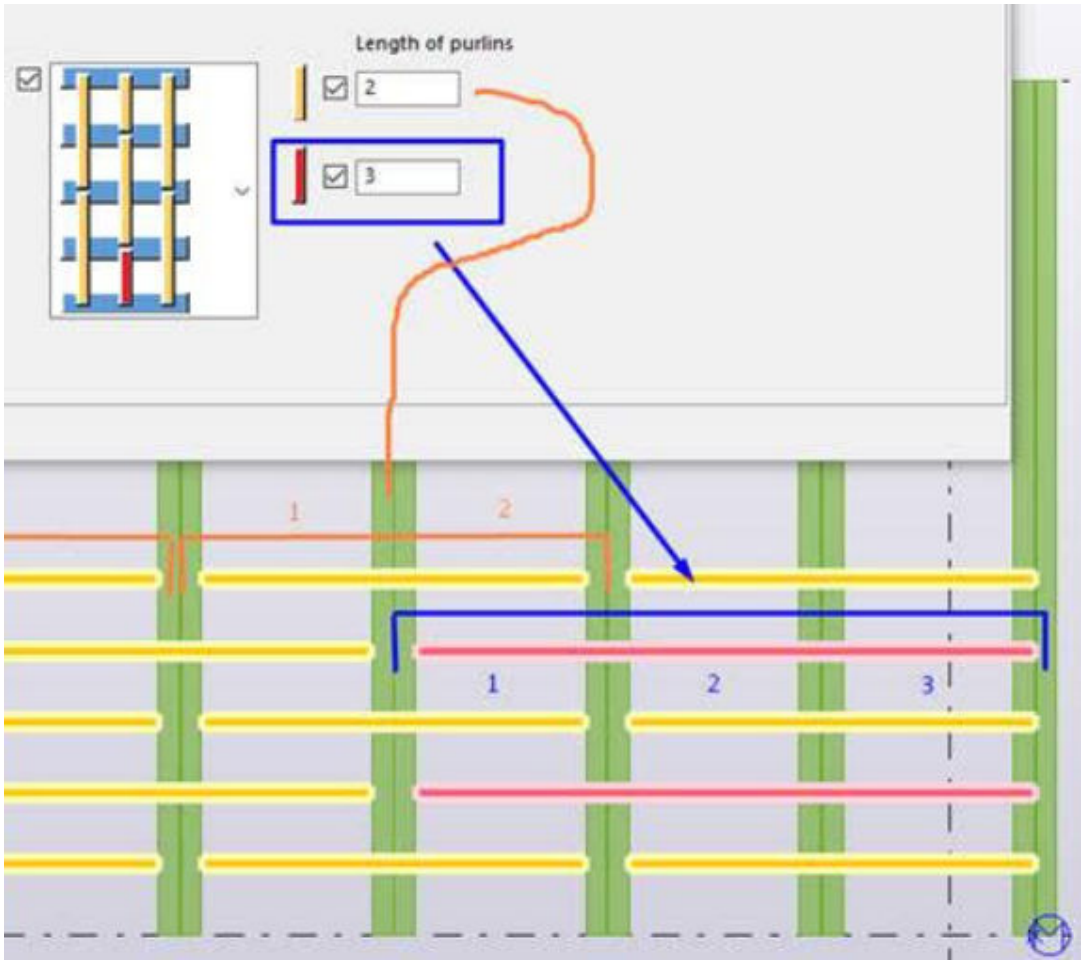
Note that if you set the overlap to **No** and the distance between the purlins is smaller than the width of the purlins, only one of the purlins is created.

Length of purlins

Define how the purlins are split. You can select both a pattern for the purlins and define how many gaps a purlin will cover.

Option	Description
	Select a pattern for the purlins from the list. The patterns with red purlins control the first purlin in odd or even purlin rows.
	Enter the number of gaps that a purlin covers in the length boxes.

For example:



Parts tab

Use the **Parts** tab to control the purlin properties and purlin positioning.

Purlin profile

Option	Description
Purlin	Define the purlin thickness, width and height. The default value is PL100*100.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
	assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Comment	Add a comment about the part.	

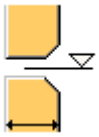
Purlin position

Option	Description
Purlins	Define the number of purlins per type. Multiple purlin types can be defined. Separate the numbers with a space. For example `1*1 4*2` results in 1 purlin of type **#1** and 4 purlins of type **#2**.
Inverse	Invert the local direction of the purlin. This is useful especially for asymmetrical profiles. You can define the inversion separately for each purlin type. The options are: `0`= direction is inverted `1`= direction is not inverted
On plane	In the first box, define the purlin position in the horizontal plane. You can define the position separately for each purlin type. The options are: `0` = middle `1` = left `2` = right In the second box, enter a value to define a horizontal offset.

Option	Description
Rotation	In the first box, enter the rotation of the purlin. You can define the rotation separately for each purlin type. The options are: 0= back 1= below 2= front 3= top In the second box, enter an angle for other rotation angles.
At depth	In the first box, define the purlin position in the vertical plane. You can define the position separately for each purlin type. The options are: 0 = middle 1 = front 2 = behind In the second box, enter a value to define a vertical offset.
Purlin panel levels	Enter one or more values to define elevations. This is useful especially for panels. Separate the numbers with a space. Use this option only if you have set the **Purlin distance according to** option to **Purlin type** on the **Picture** tab.

Elevation

Define the reference for the purlin elevations, for example, for wall panels. The elevations are set in the **Top level purlin panels** option.

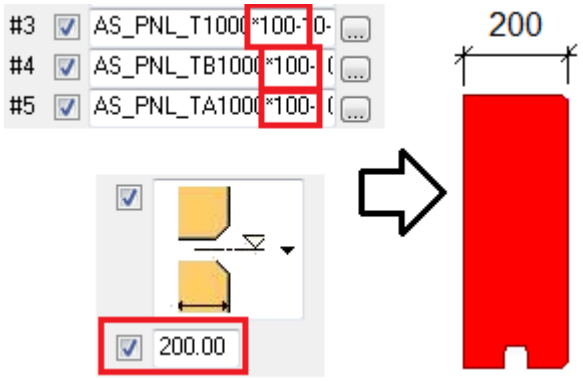
Option	Description
	Elevation is in between the parts.

Option	Description
	Elevation is on the top side of the bottom part.
	Elevation is on the bottom side of the top part.

Panel thickness

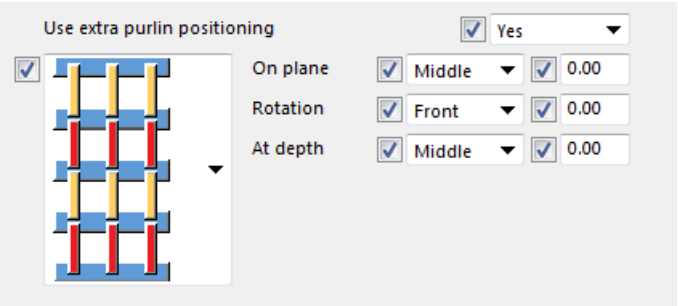
Define the thickness of the panels. This works only with parametric profiles, such as AS_PNL. Use this option only if you have set the **Purlin distance according to** option to **Purlin type** on the **Picture** tab.

The defined panel thickness overrides the width set in the purlin profile properties.

	Description	Default
Plate thickness	For example: 	100 mm

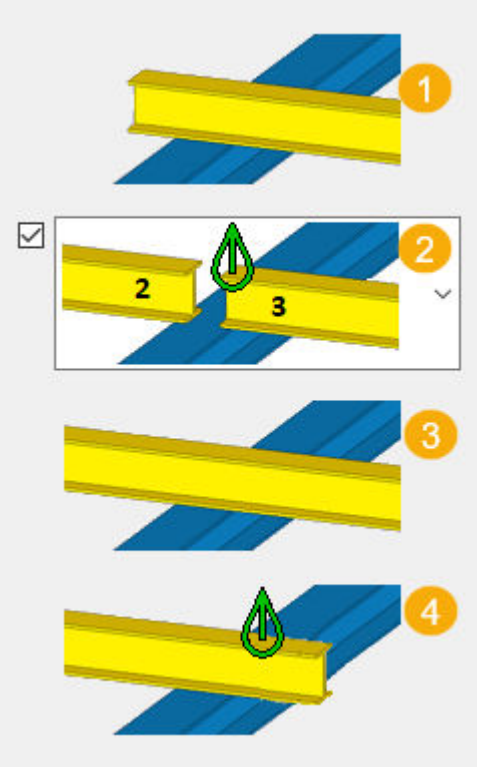
Use extra purlin positioning

Set the **Use extra purlin positioning** option to **Yes** to change the position and rotation of each even or odd purlin.



Joints tab

Use the **Joints** tab to define the components used for connecting the beams and the purlins.

Option	Description
Creation of joints	Select whether connections are created. **No**: Only the purlins are created. **Yes**: Components are added between the purlins and the beams.
Connection type  	Select the connection type from the list. You can select a suitable custom component or a connection, or you can select one of the predefined connections. You can define the connections for the start connection (**1**), middle connection with two parts (**2**), middle connection with one part (**3**), and the end connection (**4**).
Component name / number	If you have selected a custom component or a connection as the connection type, select the custom component or connection from the**Applications & components** catalog.

Option	Description
Configuration file	Select the configuration settings for the connection.
Direction, **Class**	The values you define are shown on the **General** tab of the selected connection.

UDA tab

Use the **UDA** tab to add information in the user-defined attributes (UDAs) of the parts. You can define up to 5 UDA names.

Option	Description
UDA name	Enter the name of the user-defined attribute. For example, to add a comment UDA, open the `objects.inp` file in a text editor and search for `comment`. The following attribute is shown: `attribute("comment", "j_comment", string, "%s", no, none, "0.0", "0.0")` The first text between the quotation marks is the UDA name, `comment`. The entered name is case sensitive.
Type	Select the UDA type. Use **String** for text, **Integer** for numbers, **Float** for numbers with decimals and **Option** for selecting an item in a list. You can find the UDA type in the `objects.inp` file.
Value	Enter a value for the UDA. Use text and/or numbers, depending on the defined UDA type.

Gusset+ T

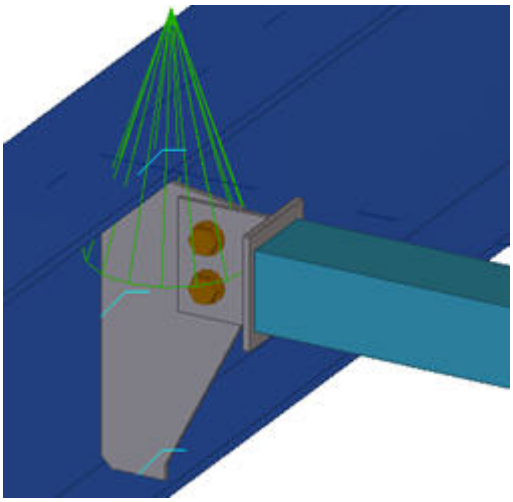
Gusset+T connects a beam or a brace to another beam by welding a T profile to the end of the beam and bolting this profile to a shear plate gusset on the main part. The main part is typically an H or I profile, and the secondary part is typically a square or round hollow profile, though channels and other profiles can also be used.

Objects created

- Stiffener
- Gusset plate
- Cut T profile (brace connection)
- Built up T profile (brace connection)

- Welds
- Bolts

Use for

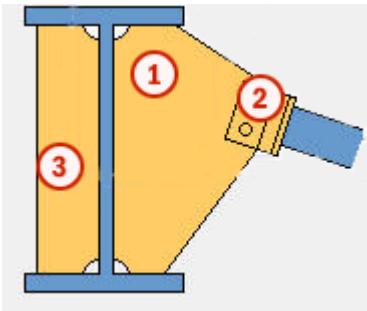
Situation	Description
	T profile welded to beam and bolted to gusset plate on main part.

Selection order

1. Select the main part (beam).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

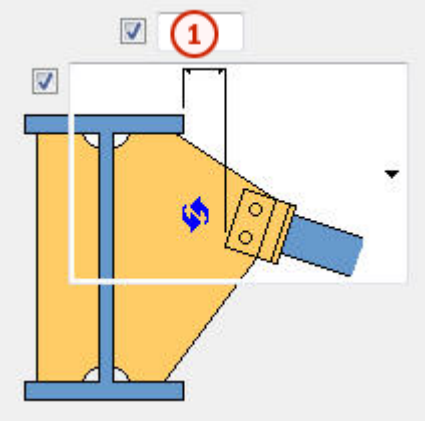


	Part
1	Gusset plate
2	Brace connection (T profile)
3	Stiffener

Picture tab

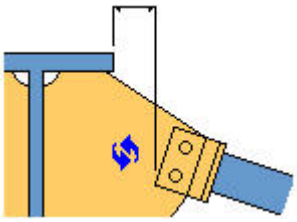
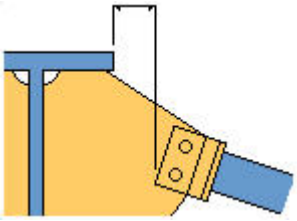
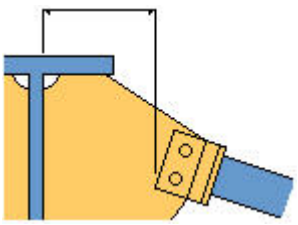
Use the **Picture** tab to control the gap and gusset plate dimensions and the shape of the gusset plate.

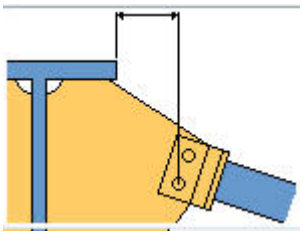
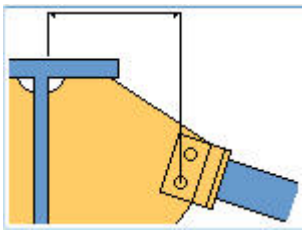
Gap dimension



	Description	Default
1	Gap dimension.	10 mm

Gap dimension options

Option	Description
	Default Gap between the edge of the main part flange and the nearest corner of the gusset plate. AutoDefaults can change this option.
	Gap between the edge of the main part flange and the nearest corner of the gusset plate.
	Gap between the face of the main part web and the nearest corner of the gusset plate.

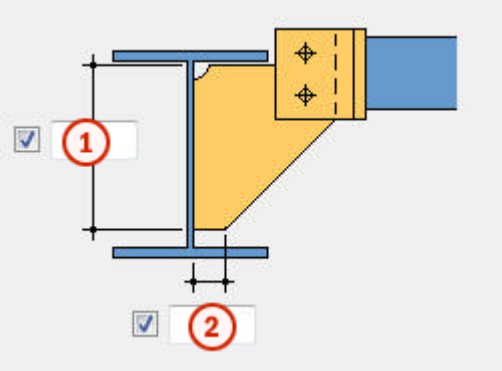
Option	Description
	Gap between the edge of the main part flange and the nearest bolt.
	Gap between the face of the main part web and the nearest bolt.

Gusset plate shape

You can further modify the gusset plate shape by defining the dimensions on the **Picture** and the **Gusset** tab.

Option	Description
	Default Regular trapezoidal gusset plate. AutoDefaults can change this option.
	Regular trapezoidal gusset plate. You can use all the gap and straight portion settings on the **Gusset** tab page to modify this gusset plate. If the gusset plate extends above the upper flange, or below the lower flange, use the extended gusset plate chamfer dimensions instead of the straight edge dimensions of the gusset plate flange.
	Square corner on the upper side of the gusset plate. You can use all the gap options to modify this gusset plate. The lower flange straight edge can also be used.
	Partial depth gusset plate. Define the height and straight edge explained in **Gusset plate dimensions**. If the gusset plate extends above the upper flange, use the extended gusset plate chamfer dimensions.

Gusset plate dimensions



	Description	Default
1	Partial gusset plate height on the web.	
2	Partial gusset plate straight edge.	`20 mm`

Gusset tab

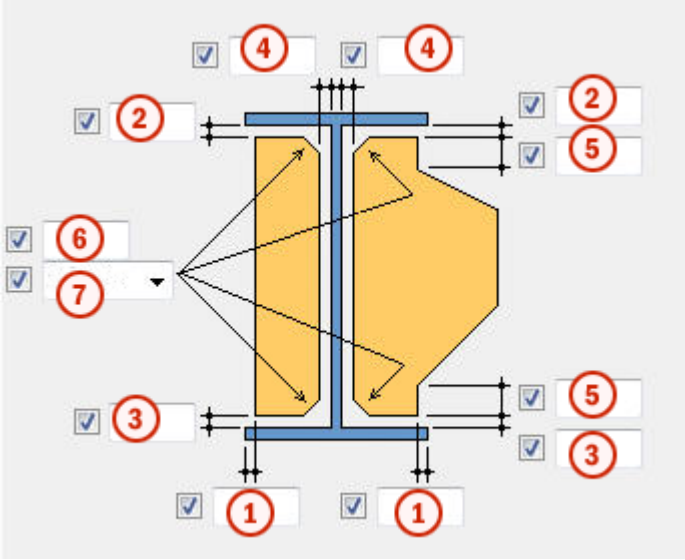
Use the **Gusset** tab to control the position and dimensions of the gusset plate and stiffener creation.

Gusset and stiffener

Option	Description	Default
Gusset	Gusset plate thickness	`6 mm`
Stiffener	Stiffener thickness	`Gusset plate thickness`

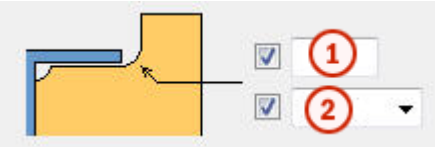
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Gusset and stiffener dimensions



	Description	Default
1	Distance of the gusset plate/stiffener from the edge of the flange.	
2	Gap between the upper flange and the gusset plate/stiffener.	
3	Gap between the lower flange and the gusset plate/stiffener.	
4	Gap between the web and the gusset plate/ stiffener.	
5	Straight portion of the gusset plate from the flange before it begins to slope to the brace.	
6	Size of the gusset plate/stiffener chamfer.	
7	Chamfer shape for the gusset plate/stiffener corners.	Line chamfer

Chamfer type and size

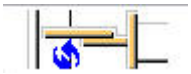


	Description	Default
1	Chamfer size for the gusset plate when the gusset plate extends above or below the main part flange.	5

	Description	Default
2	Chamfer type for the extended gusset plate.	Concave arc chamfer

	Description	Default
1	Chamfer size for the square corner on the upper side of the gusset plate.	20
2	Chamfer type for the square corner.	Concave arc chamfer

Gusset position

Option	Description
	Default Gusset plate above the Cut T profile. AutoDefaults can change this option.
	Gusset plate above the Cut T profile.
	Gusset plate below the Cut T profile.

Stiffener creation

Option	Description
	Default No stiffener. AutoDefaults can change this option.
	Stiffener perpendicular to the main part.
	Stiffener aligned with the gusset.
	No stiffener.

Brace connection tab

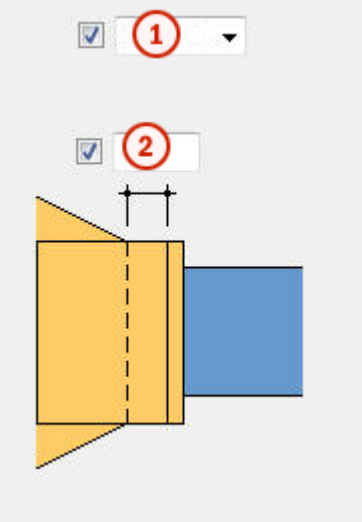
Use the **Brace connection** tab to define how the brace is connected to the gusset plate.

Profile properties

Option	Description	Default
Cut T	Cut T profile thickness, width and height by selecting the profile from the profile catalog.	`Gusset plate thickness`
Flange	Flange thickness for the Built up T profile. Note that you must first define the profile type to be **Built up T**.	`6 mm`
Web	Web thickness for the Built up T profile. Note that you must first define the profile type to be **Built up T**.	`6 mm`

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Brace connection profile

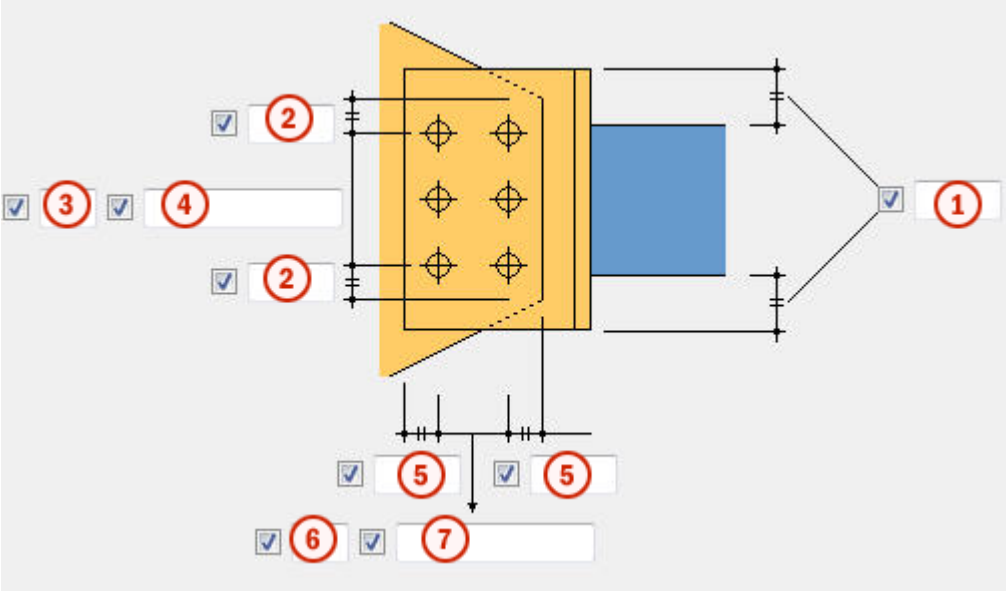


	Description	Default
1	Type of the T profile to be used in the brace connection. When you select **Built up T**, also define the gap between the gusset plate and the Built up T profile.	
2	Gap between the gusset plate edge and the flange of the Built up T profile.	10 mm

Bolts tab

Use the **Bolts** tab to control the properties of bolts that connect the gusset plate to the T profile.

Bolt group dimensions



	Description	Default
1	Minimum T profile extension for the top and bottom of the brace connection.	10 mm
2	Bolt edge distance.	40 mm
3	Number of bolts.	2
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.	60 mm
5	Bolt edge distance.	40 mm
6	Number of bolts.	1
7	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.	60 mm

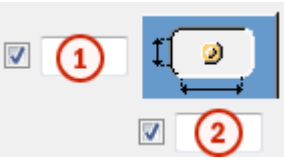
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.

Option	Description	Default
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

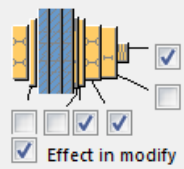


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Main part welds / Cut T welds tabs

Use the **Main part welds** and the **Cut T welds** tabs to define the weld properties. You can select whether continuous, chain intermittent, or staggered intermittent weld shapes are used.

Click the link below to find out more:

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

5.10 Tubes

This section introduces components that can be used in steel tube connections.

Click the links below to find out more:

- Tube splice (6) (page 1984)
- Tube gusset (20) (page 1992)
- Squeezed tube bolted (102) (page 2020)
- Squeezed tube (103) (page 2033)
- Tube-Chamfer (page 2040)
- Tube-CrossingSaddle (page 2042)
- Tube-MitreSaddle+Hole (page 2044)
- Tube-Saddle+Hole (page 2046)
- Tube-SlottedHole (page 2050)

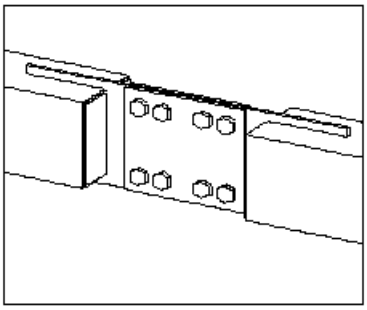
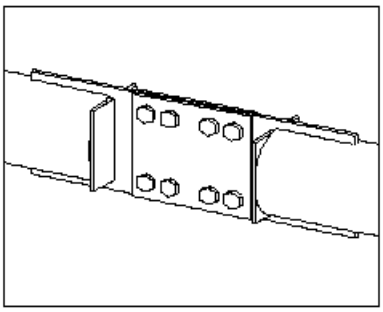
Tube splice (6)

Tube splice (6) connects two rectangular hollow core sections or round tubes with splice plates. End plates are created at both ends of the connection. Connection plates create cuts into the connected parts.

Objects created

- Splice plates
- Connection plates
- End plates
- Welds
- Bolts
- Cuts

Use for

Situation	Description
	Tube splice connection to rectangular hollow core sections.
	Tube splice connection to round tubes.

Selection order

1. Select the main part (column or beam).
2. Select the secondary part (column or beam).

   The connection is created automatically when the secondary part is selected.

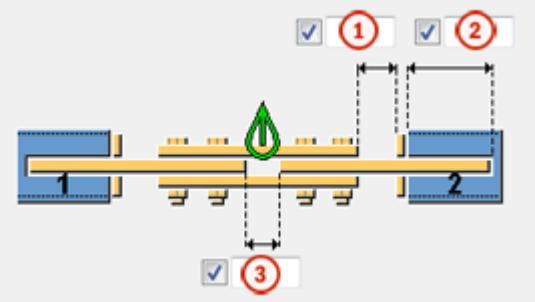
Part identification key

	Part
1	Splice plate
2	Connection plate
3	End plate

Picture tab

Use the **Picture** tab to control the positions of the plates.

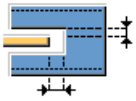
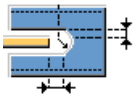
Plate position



	Description	Default
1	Distance between the splice plate and the end plate.	10 mm
2	Connection plate cut depth. The cut depth affects the size of the connection plate.	150 mm
3	Distance between the connection plates.	20 mm

Cut creation

Define whether connection plates create cuts to the parts they connect, and whether end plates are cut.

Option	Description
	Default Connection plate does not create a cut. AutoDefaults can change this option.
	Connection plate does not create a cut.
	Connection plate creates a rectangular cut. Define the horizontal and vertical cut dimensions.
	Connection plate creates a round cut. Define the horizontal and vertical cut dimensions, and the radius of the cut.

Option	Description
	Default End plate is not cut. AutoDefaults can change this option.
	End plate is not cut.
	End plate is cut.

Parts tab

Use the **Parts** tab to control the size, position, material, name and finish of the plates.

Plate

Opttion	Description
Splice plate	Splice plate thickness.
Connection plate	Connection plate thickness.
End plate	End plate thickness.

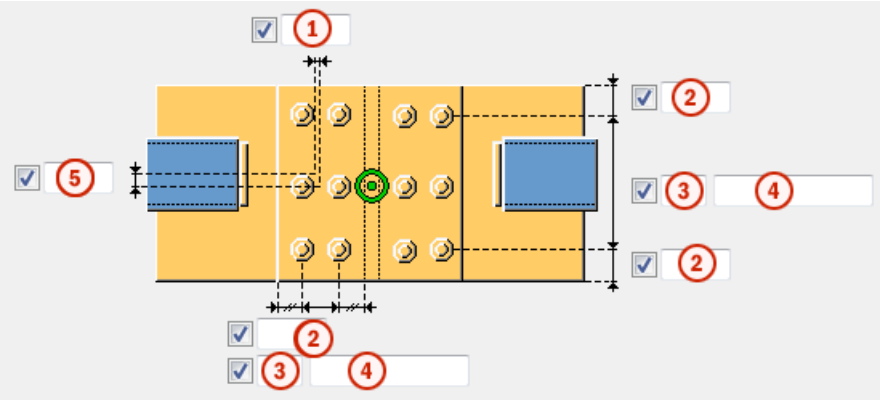
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Option	Description	Default
Finish	Describes how the part surface has been treated.	

Bolts tab

Use the **Bolts** tab to control the properties of the bolts that connect the splice plates and the connection plates.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for the bolt group position from the center line of the connected parts.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.

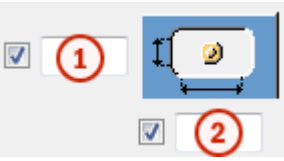
Option	Description
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

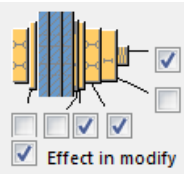


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

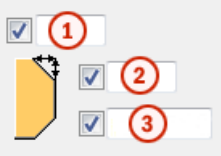
End plates tab

Use the **End plates** tab to control the shape and the dimensions of the end plate.

End plate shape

Option	Description
	Square Default
	Square
	Round

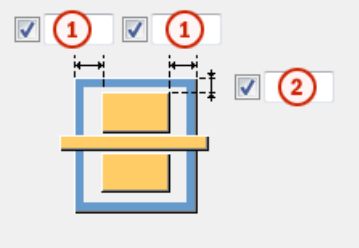
End plate chamfer dimensions



	Description
1	Horizontal chamfer dimension.
2	Vertical chamfer dimension.
3	Select the chamfer type.

End plate dimensions

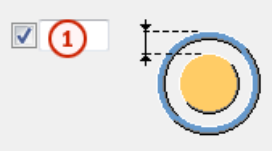
Define the dimensions of the square end plate.



	Description
1	Horizontal dimension from the edge of the end plate to the flange of the column or the beam.
2	Vertical dimension from the edge of the end plate to the flange of the column or the beam.

End plate dimensions

Define the dimension of the round end plate.



	Description
1	Dimension from the edge of the end plate to the outer edge of the column or the beam.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Tube gusset (20)

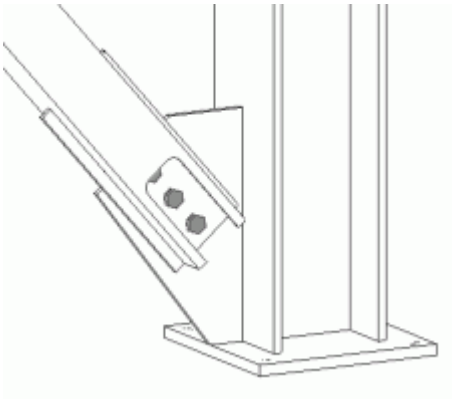
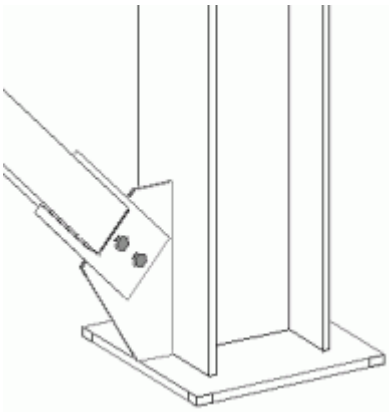
Tube gusset (20) connects 1 to 10 hollow braces to a beam or a column using a gusset plate. The braces need to have a rectangular hollow section (RHS) or a tube profile. The connection bolts the braces to the gusset plate using a connection plate and an optional tongue plate. The hollow braces can be sealed with end plates.

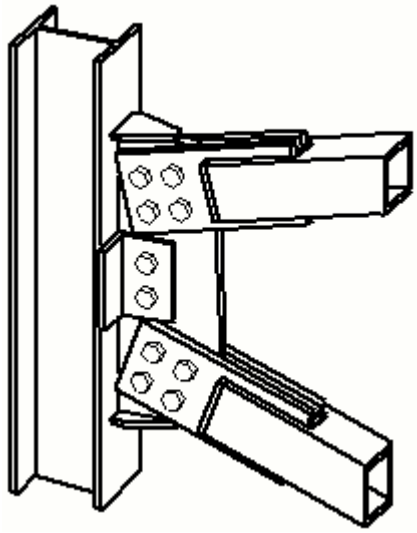
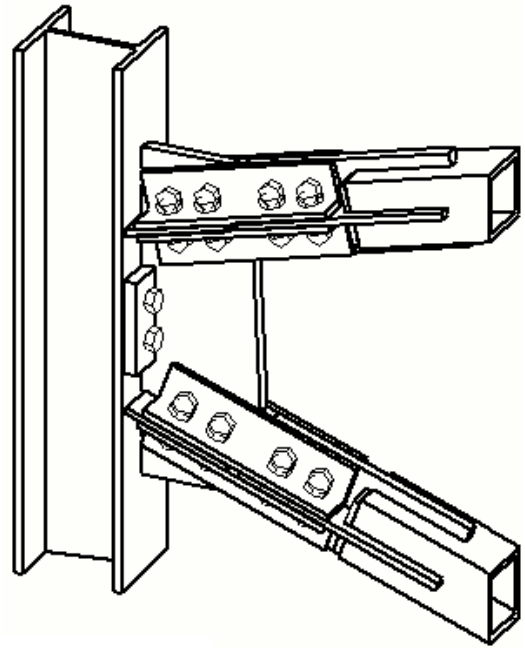
Objects created

- Gusset plate
- Connection plates
- Clip angles
- End plates (seal plates)

- Tongue plates
- Cover plates
- Stiffeners
- Bolts
- Welds

Use for

Situation	Description
	Gusset plate is welded to column flange. Brace is welded to connection plate. The end of the brace is notched to accommodate the bolts in the connection between the connection plate and the gusset plate.
	Gusset plate is welded to column flange. Brace is bolted to gusset plate using a tongue plate.

Situation	Description
	Gusset plate is connected to column flange with clip angles. Brace is bolted to gusset plate using a tongue plate. The braces are sealed with end plates, and stiffeners are created.
	Gusset plate is connected to column flange with a connection plate. Cross plates and cover plates are created.

Before you start

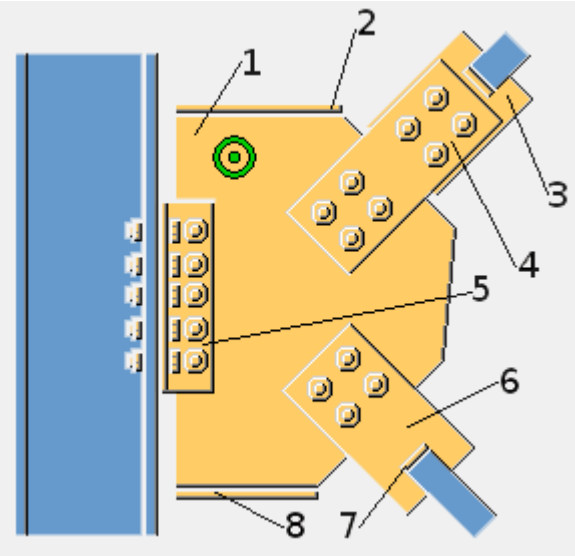
Create a beam or a column and 1 to 10 braces with RHS or tube profile.

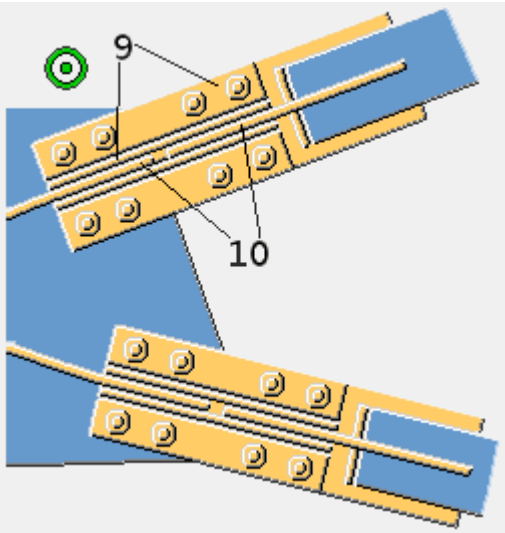
Selection order

1. Select the main part (column or beam).
2. Select the secondary part (first brace).
3. Select the second secondary part (second brace).
4. Select the subsequent secondary parts (subsequent braces).
5. Click the middle mouse button to create the connection.

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Part identification key





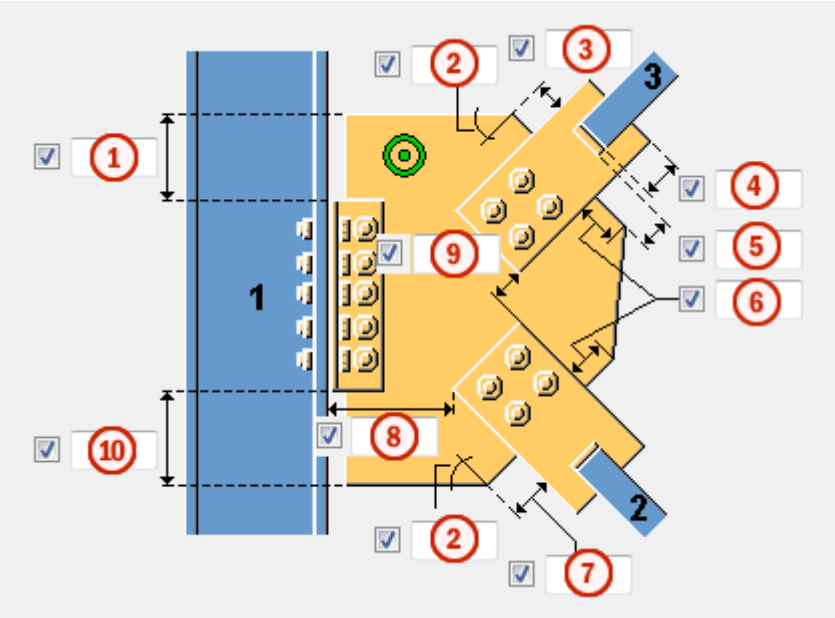
	Part
1	Gusset
2	Stiffener 1
3	Tongue plate
4	Cover plate Created on the **Brace conn** tab.
5	Clip angle

	Part
6	Connection plate Created on the **Brace conn** tab.
7	End plates (seal plates)
8	Stiffener 2
9	Cover plate Created on the **Cross plates** tab.
10	Cross plate

Picture tab

Use the **Picture** tab to control the gusset plate dimensions and positioning.

Dimensions



	Description	Default
1	Distance between the clip angle or connection plate upper edge and the gusset plate upper edge.	
2	Corner angle of the gusset plate (in degrees). This value affects the gusset plate shape.	
3	Length of the edge of the gusset plate that is perpendicular to the uppermost brace. This value affects the gusset plate shape.	

	Description	Default
4	Brace length on the connection plate. Enter a negative value to prevent the connection plate from being inside the brace.	150 mm
5	Distance between the gusset plate and the brace. If the braces are sealed with end plates, the distance is between the gusset plate and the end plate.	20 mm
6	Length of the edges of the gusset plate perpendicular to the braces. This value affects the gusset plate shape.	
7	Length of the edge of the gusset plate that is perpendicular to the lowest brace. This value affects the gusset plate shape.	
8	Distance between the main part and the first picked brace.	
9	Distance between the braces.	
10	Distance between the clip angle or the connection plate lower edge and the gusset plate lower edge.	

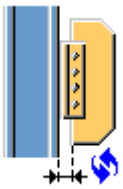
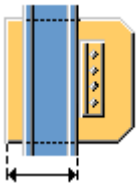
NOTE The following examples show only some of the available options. You will find more options on the **Picture** tab.

Gusset plate positioning

Define how the gusset plate is positioned when a base plate is used.

Option	Description
	Default Gusset plate is parallel to the main part. AutoDefaults can change this option.
	Gusset plate is parallel to the brace.

Gusset plate dimensions

Option	Description
	Default Gusset plate does not run through the main part. Define the gusset plate cut dimension. AutoDefaults can change this option.
	Gusset plate does not run through the main part. Define the gusset plate cut dimension.
	Gusset plate runs through the main part. Define the gusset plate extension dimension.

Gusset tab

Use the **Gusset** tab to control the gusset plate properties, shape and position, and clip angle properties and orientation.

Plates

Option	Description	Default
Gusset	Gusset plate thickness, width and height.	
Connection plates	Connection plate thickness and width.	`no connection plate is created`
L profile	Clip angle profile by selecting it from the profile catalog.	`L100*100*10`

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
	where you can enter the assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

NOTE The following examples show only some of the available options. You will find more options on the **Gusset** tab.

Gusset plate connection

Define how the gusset plate is connected to the main part.

Option	Description
	Default Gusset plate is welded directly to the main part. AutoDefaults can change this option.
	Gusset plate is connected to the main part with clip angles. Select to which side of the gusset plate the clip angles are created.
	Gusset plate is connected to the main part with a connection plate. Select to which side of the gusset plate the connection plate is created.

Clip angle orientation

Define how the clip angle is placed on the connection.

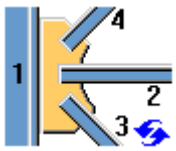
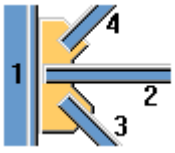
Option	Description
	Default Clip angle is placed on the connection so that the longer leg is connected to the gusset plate. AutoDefaults can change this option.
	Clip angle is placed on the connection so that the longer leg is connected to the main part.

Gusset plate shape

When you select the option to optimize the gusset plate weight, you can define whether the selection order of the braces affects the position of the braces.

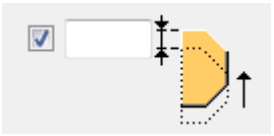
Option	Description
	Default AutoDefaults can change this option.
	This option optimizes the gusset plate weight.

Brace position

Option	Description
	Default The brace position is not affected. AutoDefaults can change this option.
	The first selected brace is placed closest to the main part.

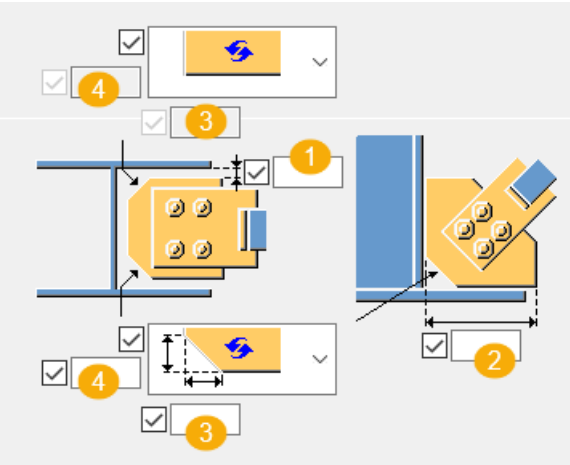
Gusset plate position on the brace

Define where to place the gusset plate on the brace. If needed, you can fine-tune the gusset plate position by moving the gusset plate in the z or in the y direction.

Option	Description
	Default Gusset plate is positioned in the middle of the brace. AutoDefaults can change this option.
	Gusset plate is positioned on the top flange of the brace.
	Define how much the gusset plate is moved in the z direction.
	Define how much the gusset plate is moved in the y direction.

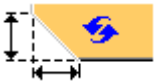
Gusset plate chamfer

Define the gusset plate chamfer type and dimensions.



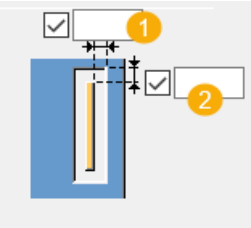
	Description	Default
1	Distance between the connection plate and the inner flange of the main part.	
2	Horizontal distance between the gusset plate edge and the flange of the main part.	
3	Horizontal dimension of the chamfer. By default, the second chamfer is not created.	`10 mm`
4	Vertical dimension of the chamfer. By default, the second chamfer is not created.	`10 mm`

Define the chamfer type.

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	Convex arc chamfer
	Concave arc chamfer

Cut size

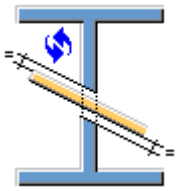
If the gusset plate runs through the main part, define the size of the cut created for the gusset plate.

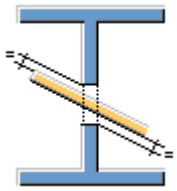


	Description
1	Define the horizontal size of the cut.
2	Define the vertical size of the cut.

Cut options

If you use beam-column-beam connections and want the gusset plate to create a cut, you can define how the cut runs through the main part.

Option	Description
	Default Cut is created according to the gusset plate orientation. AutoDefaults can change this option.

Option	Description
	Straight cut.

Brace conn tab

Use the **Brace conn** tab to control connection plate, tongue plate, and end plate properties .

Brace connection

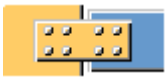
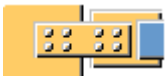
Option	Description	Default
Connection plate	Thickness, width and height of the connection plate.	`thickness = 20 mm`
End plates	Thickness, width and height of the end plate.	`thickness = 5 mm`
Middle end plate	Thickness and height of the middle end plate.	`no middle end plate is created`
Tongue plate	Thickness and height of the tongue plate.	`no tongue plate is created`
Cover plate	Thickness, width and height of the cover plate.	`no cover plate is created`
Stiffener	Thickness, width and height of the stiffener.	`no stiffener is created`

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

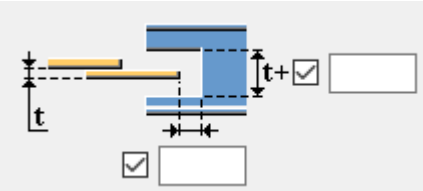
Brace connection types

Define how the brace is connected to the connection plate.

Option	Description
	Default Brace is welded AutoDefaults can change this option.
	Brace is welded.
	Brace is bolted.
	Brace is welded and notched around the nuts.
	Tongue plate and cover plate are created.

Cut in bracing

If needed, you can create a cut in the bracing.



Define the width of the cut in the bracing, where **t** is the thickness of the connection plate.

Define the length of the cut in the bracing from the edge of the connection plate.

Gusset and connection plate distance

Define the distance between the gusset plate and the connection plate.

Round cut in bracing

If needed, you can create a round cut in the bracing. Enter the radius value.

NOTE The following examples show only some of the available options. You will find more options on the **Brace conn** tab.

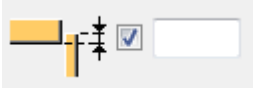
Connection plate

Define whether the brace is notched or the connection plate cut when the connection plate is connected to the brace.

Option	Description
	Default Brace is notched. AutoDefaults can change this option.
	Connection plate is cut.
	Connection plate is cut, but the part of the connection plate created inside the bracing is not deleted.
	If you cut the connection plate, you can define the size of the gap between the brace and the connection plate.

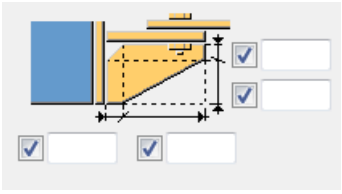
Number of connection plates

Define whether one or two connection plates are used for connecting the brace to the gusset plate.

Option	Description
	Default One connection plate. AutoDefaults can change this option.
	Two connection plates and a middle end plate at the ends of the connection plates.
	If a middle end plate is created, you can define the width of the end plate.

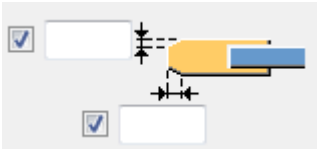
Option	Description
	Select the middle end plate position.

Connection plate stiffeners

Option	Description
	To create stiffeners, define the stiffener thickness. By default, one stiffener is created. When you select to create two connection plates, you can also select to create one stiffener on the left or one on the right, or one stiffener on both the left and the right side.
	Define the inner and outer chamfers of the stiffeners.

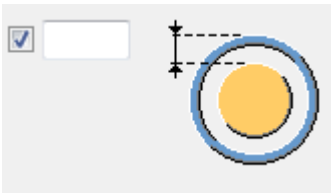
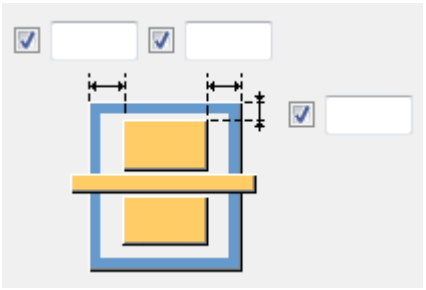
Connection plate chamfer

Define whether the connection plate is chamfered.

Option	Description
	Default No chamfers are created. AutoDefaults can change this option.
	Chamfers are created.
	If you create chamfers, define the vertical and horizontal chamfer dimensions.

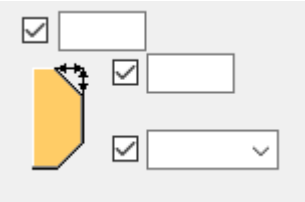
End plates

If you use the end plates to seal the braces, define the end plate shape and dimensions.

Option	Description
	Default Square end plate. AutoDefaults can change this option.
	Round end plate.
 	End plate edge distance from the brace outer edge.

End plate chamfer

Define the end plate chamfer type and horizontal and vertical dimensions.



Stiffeners tab

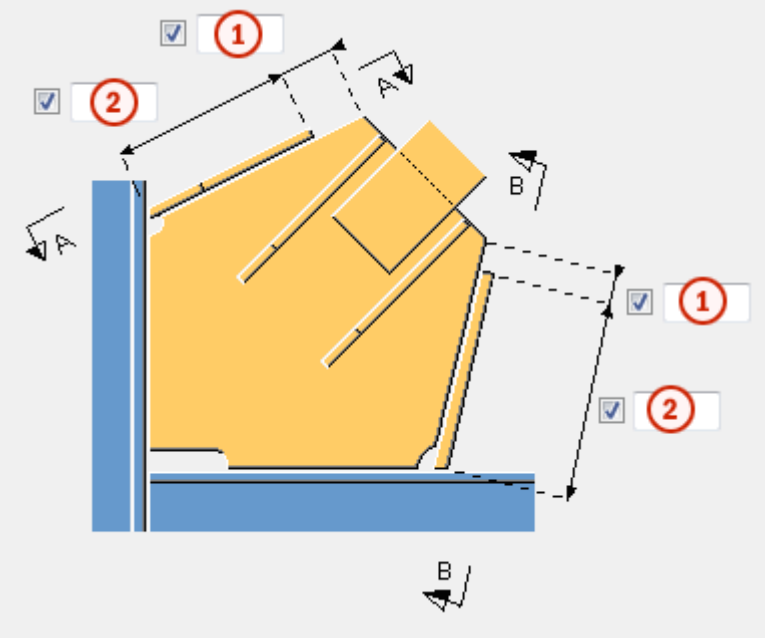
Use the **Stiffeners** tab to control the stiffener properties and dimensions.

Stiffeners

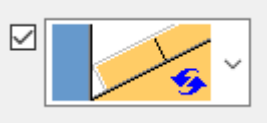
Option	Description	Default
Stiffener 1 **Stiffener 2**	Stiffener thickness.	`no stiffener is created`

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Stiffener length

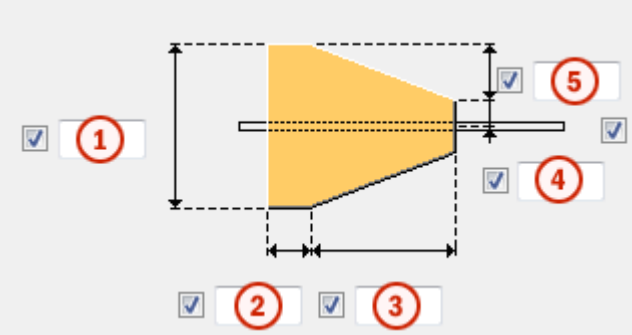


	Description
1	Distance between the stiffener edge and the gusset plate edge.
2	Length of stiffener.



You can fit the stiffeners to the main part. By default, the stiffeners are not fitted.

Stiffener dimensions

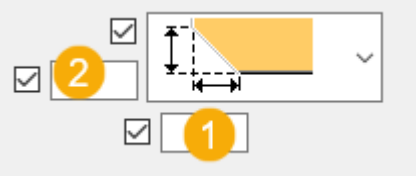


	Description
1	Width of the stiffener.
2	Length of the stiffener base.
3	Length of the skew part of the stiffener.
4	Distance from the stiffener center line.
5	Vertical distance between the stiffener base and the skew part.

Chamfer type

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Chamfer dimensions

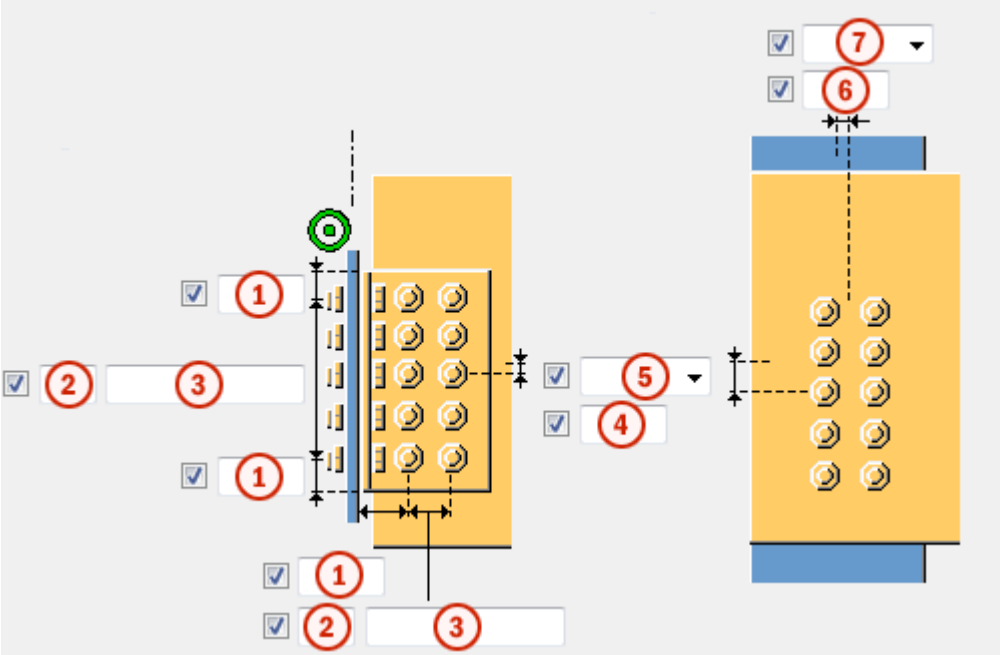


	Description
1	Horizontal dimension of the chamfer.
2	Vertical dimension of the chamfer.

Gusset conn tab

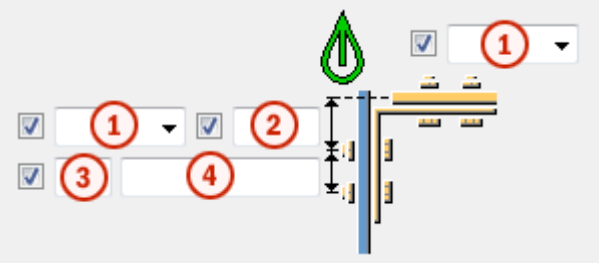
Use the **Gusset conn** tab to control the bolt group properties for bolts that connect the gusset plate to the main part, and to control the clip angle attachment.

Bolt group dimensions on gusset plate



	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.

	Description
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Dimension for vertical bolt group position.
5	Select how to measure the dimensions for vertical bolt group position.
6	Dimension for horizontal bolt group position.
7	Select how to measure the dimensions for horizontal bolt group position.



	Description
1	Location where the bolts should be attached.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

NOTE The following examples show only some of the available options. You will find more options on the **Gusset conn** tab.

Clip angle attachment type

Define how the clip angle is attached to the gusset plate and to the main part.

Option	Description
	Default Both parts are bolted. AutoDefaults can change this option.
	Automatic When the main part is a tube profile, the clip angles are welded to the main

Option	Description
	part and bolted to the secondary part. Otherwise the clip angles are bolted to both parts.
	Main part is bolted and secondary part is welded.
	Main part is welded and secondary is part bolted.
	Both parts are bolted.
	Both parts are welded.

Bolts on gusset plate

Define whether the gusset plate is connected to the main part with bolts when no clip angles are used.

Option	Description
	Default Bolts are not created in the gusset plate. AutoDefaults can change this option.
	Bolts are created in the gusset plate.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3

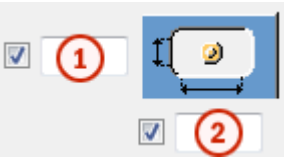
Option	Description
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



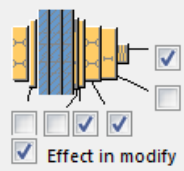
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

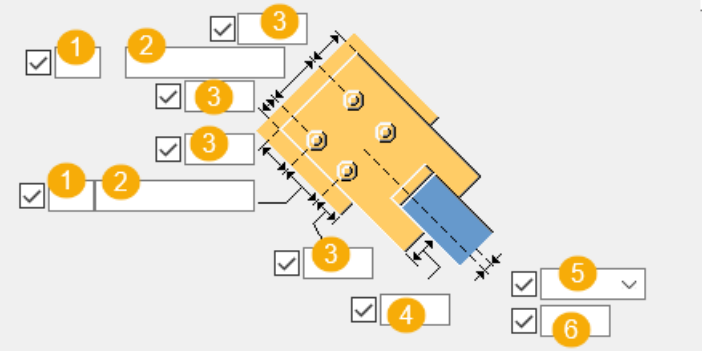
Bolt length increase

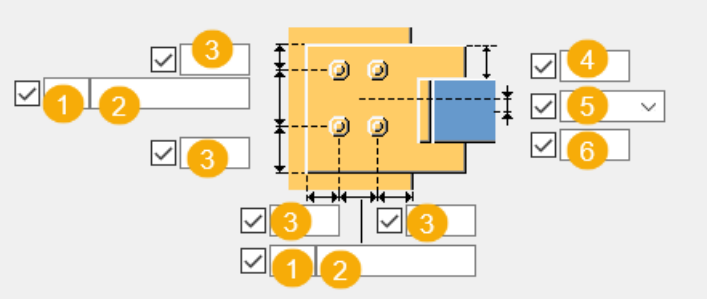
Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

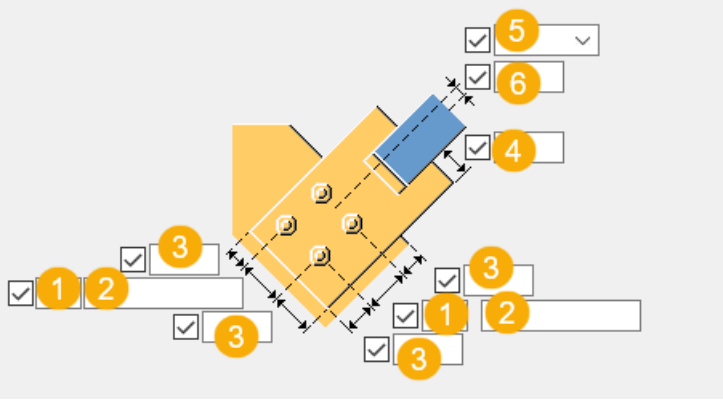
Brace bolts 1/Brace bolts 2/Brace bolts 3 tab

Use the **Brace bolts 1**, **Brace bolts 2** and **Brace bolts 3** tabs to control the bolts that connect the first, the second, and the subsequent braces to the gusset plate.

Bolt group dimensions on connection plates



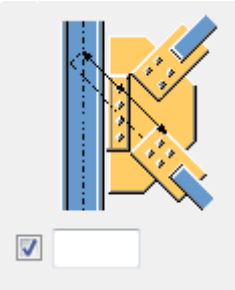
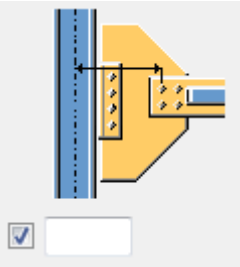
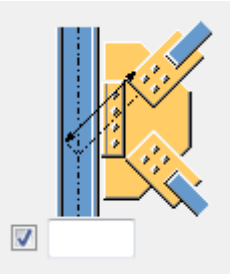




	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Distance between the brace and the connection plate edge.
5	Select how to measure the dimensions for vertical bolt group position.
6	Dimension for vertical bolt group position.

Bolt distance

Define the minimum distance from the connection plate bolts to the intersection point of the main part and brace center lines. If a brace is perpendicular to the main part, the distance is measured from the main part center line to the nearest bolts.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

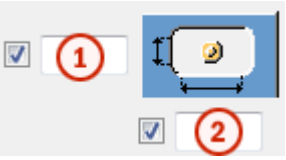
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.

Option	Description	Default
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

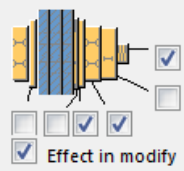
Bolt type

Select the bolt type to define the location where the bolts should be attached.

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Cross plates tab

Use the **Cross plates** tab to control the cross plate and the cover plate properties and position.

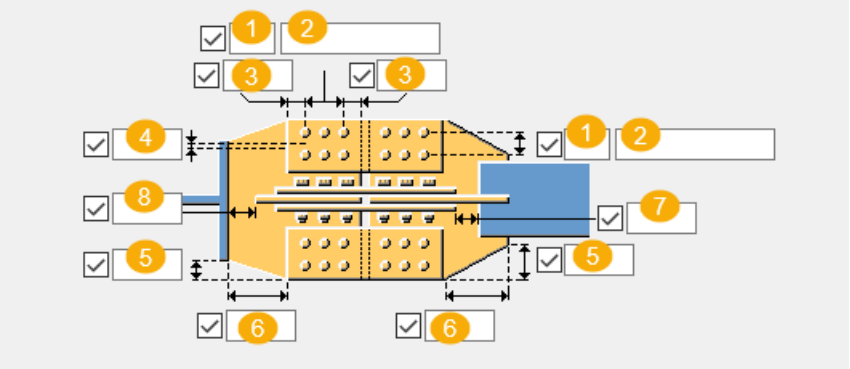
Cross plates

Option	Description	Default
Cross plate	Thickness, width and height of the cover plate.	`no cross plate is created`
Cover plate	Thickness, width and height of the cover plate.	`no cover plate is created`

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Bolt group dimensions



	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Dimension for the horizontal bolt group position.
5	Vertical dimension of the chamfer.
6	Horizontal dimension of the chamfer.
7	Distance between the end of the brace cap plate and the edge of the splice plate.
8	Clearance of the cross plate edge on the gusset plate from the surface of the main part.

Cross and cover plate position

Define the position of the cross plate and the cover plate.

Option	Description
	Default Cover plate is created on both sides of the cross plate. AutoDefaults can change this option.
	Cover plate is created on top of the cross plate.
	Cover plate is created on the bottom of the cross plate.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Squeezed tube bolted (102)

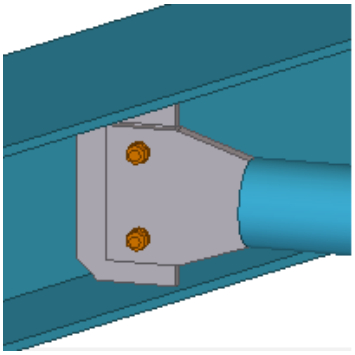
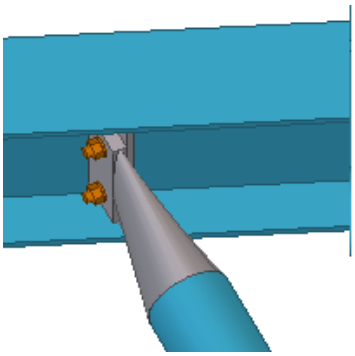
Squeezed tube bolted (102) creates a squeezed part between the main part and a tubular profile. The main part must be an I or H profile. The squeezed part can either be a tube that is squeezed at one end and then welded to a plate, or a contour plate.

Objects created

- Squeezed tube, or reducing contour plate
- Stiffeners
- Bolts

- Welds

Use for

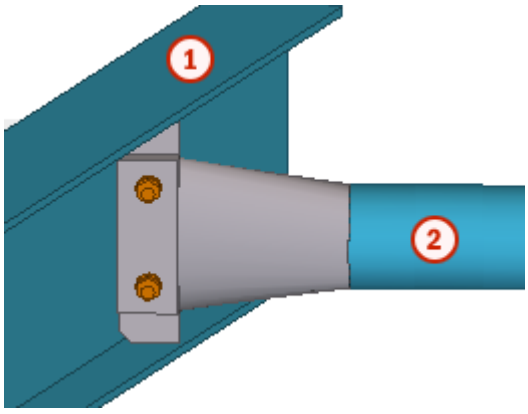
Situation	Description
	A tubular profile is welded to a bracing which is bolted to a gusset plate. The gusset plate is welded to the main part.
	A simplified tensioner profile.

Selection order

1. Select the main part (I or H profile).
2. Select the secondary part (tubular profile).

   The squeezed part is created automatically.

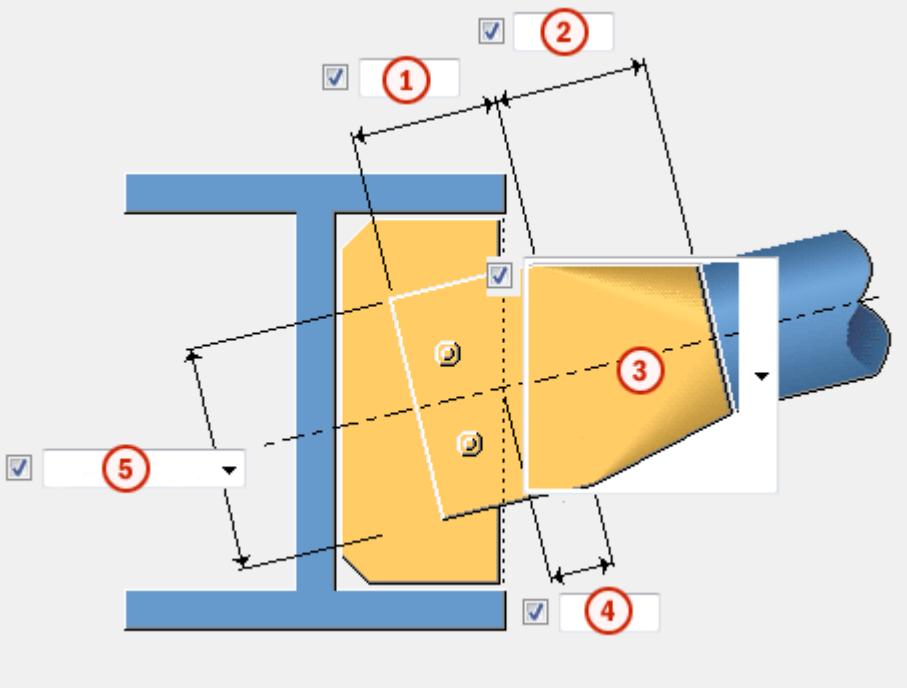
Part identification key



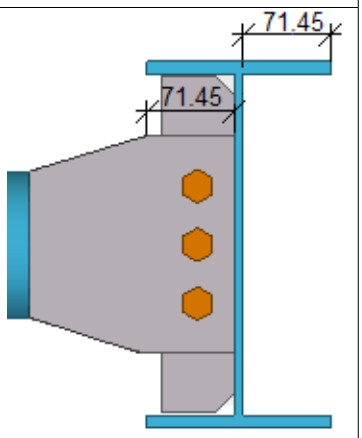
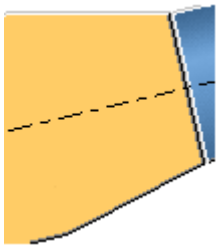
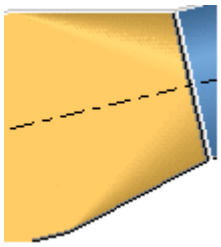
	Part
1	Main part (I profile)
2	Tubular profile

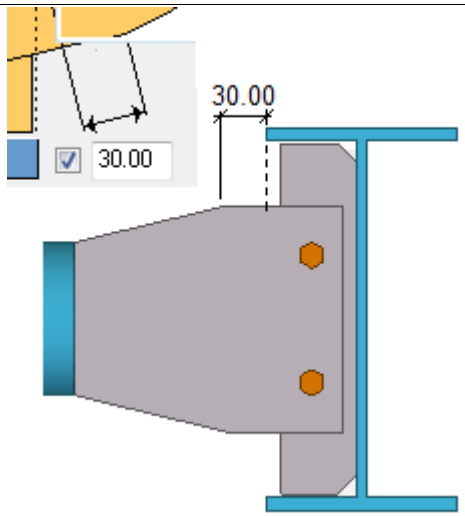
Picture tab

Use the **Picture** tab to control the bracing levels and offsets.



	Description	Default
1	Width of the squeezed part extension.	Depends on main part: `main part width - web thickness / 2.` Example:

	Description	Default
		
2	Width of the squeezed part.	120 mm
3	Select the type of the reducing part. Plate:  Squeezed tube: 	
4	Horizontal edge offset. Example:	0 mm

	Description	Default
		
5	Select how the height of the squeezed part extension is calculated.	

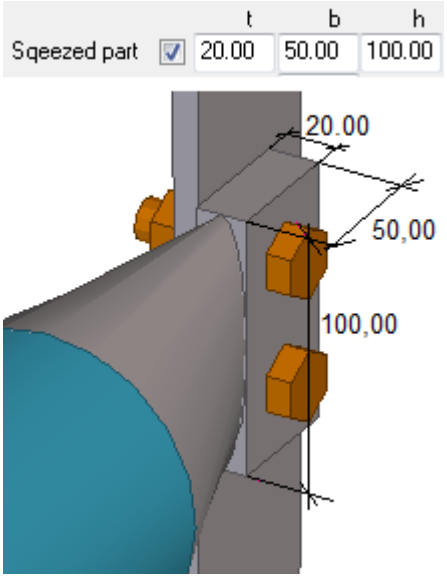
Parts tab

Use the **Parts** tab to control the dimensions of the squeezed part and the stiffener properties.

Squeezed part

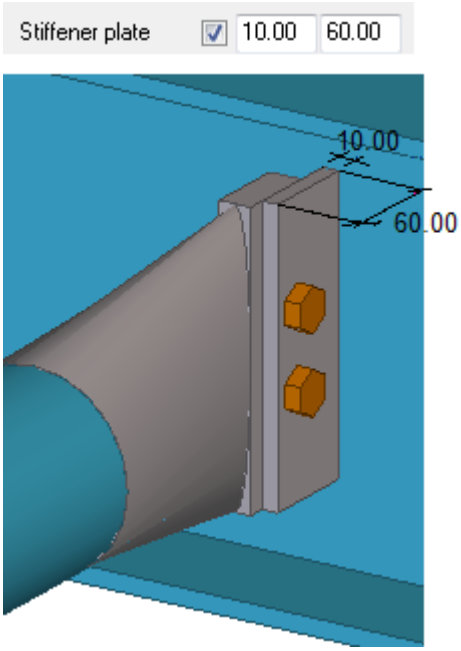
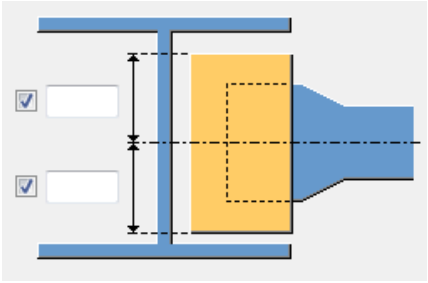
Define the dimensions and properties of the squeezed part extension.

Option	Description
Squeezed part	Thickness, width and height of the squeezed part extension. Example:

Option	Description
	 Use the second row to define the width of the squeezed part. The width defined on the **Parts** tab overrides the width defined on the **Picture** tab.
Parts add	Select whether the squeezed parts are handled as loose parts, meaning that they are not attached to any other profile in the component, or whether they are added to the secondary part. • **Yes** Squeezed parts are part added to the tubular profile. • **No** Squeezed parts remain loose parts.

Stiffeners

Option	Description
Stiffeners are	Select whether the stiffeners are created as stiffeners or as plates. • **Stiffeners** Define the stiffener dimensions on the **Stiffeners** tab.

Option	Description
	• **Plate stiffeners** Define the dimensions using the **Stiffener plate** boxes.
Stiffener plate	Define the thickness and width of the stiffener plate. **NOTE** You cannot define the plate height. The height is equal to squeezed part height defined on the **Picture** tab. 
	Define the stiffener plate height above and below the tubular profile center line. These values only take effect if you have set the **Stiffeners are** option to **Plate stiffeners**.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
	assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

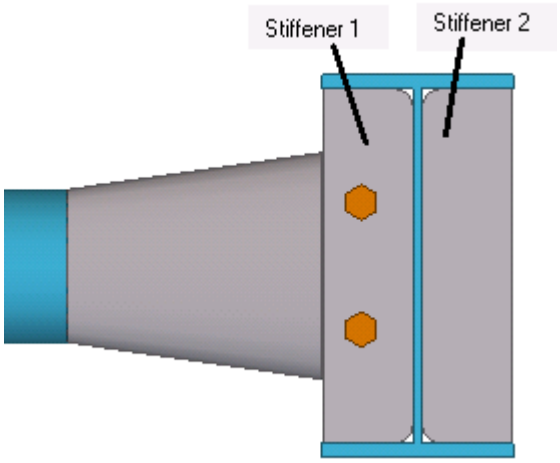
Stiffeners tab

Use the **Stiffeners** tab to control the stiffener properties.

Stiffener 1, Stiffener 2

NOTE The options on this tab work only if you have set the **Stiffeners are** option to **Stiffeners** on the **Parts** tab.

Stiffener 1 is the stiffener on the side of the squeezed tube. **Stiffener 2** is the stiffener on the other side of the web.



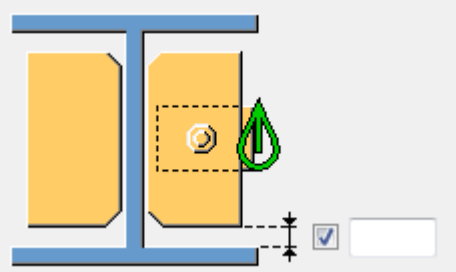
Define the thickness, width and height of the stiffeners.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

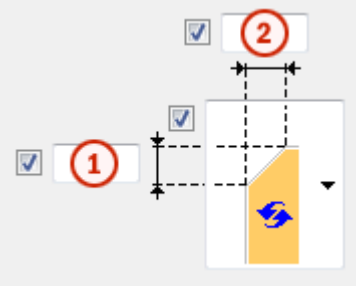
Option	Description	Default
	assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Stiffener gap

Define the size of the gap between the beam flange and the stiffener.

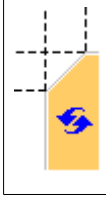


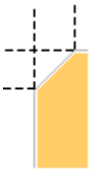
Chamfer dimensions



	Description
1	Vertical dimension of the chamfer.
2	Horizontal dimension of the chamfer.

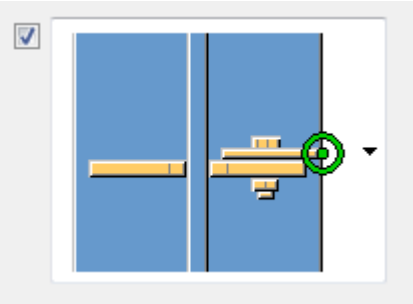
Chamfer type

Option	Description
	Default. Line chamfer AutoDefaults can change this option.

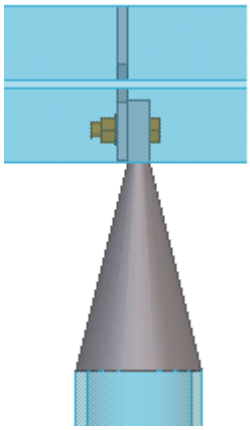
Option	Description
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Stiffener side

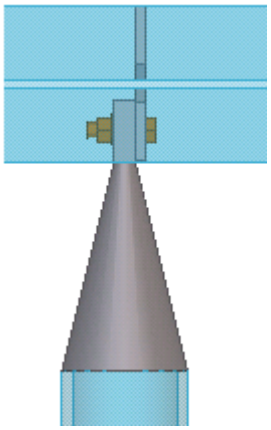
Define the side of the stiffeners.



On the left side of the squeezed part:



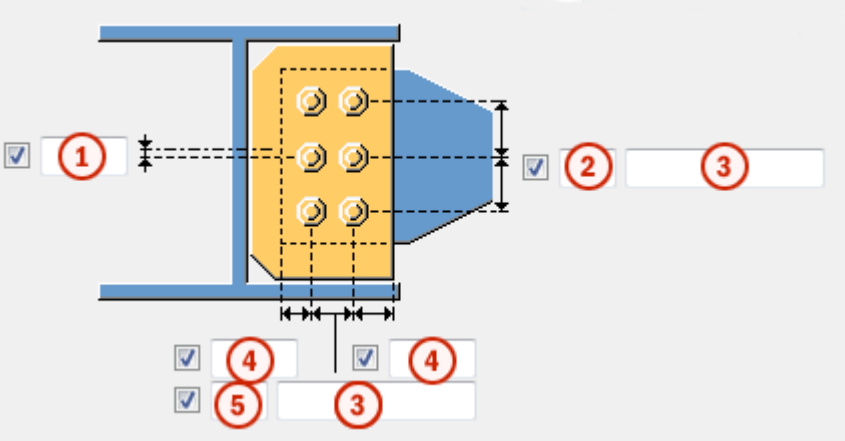
On the right side of the squeezed part:



Bolts tab

Use the **Bolts** tab to control the bolts.

Bolt group dimensions



	Description	Default
1	Bolt vertical offset from the center line.	0 mm
2	Number of bolts in vertical direction.	2
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.	
4	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.	55 mm
5	Number of bolts in horizontal direction.	1

Bolt group orientation

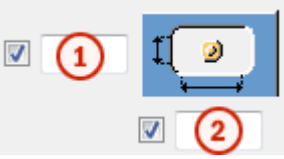
Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Staggered Bolts are staggered in the direction of the secondary part.
	Square Square bolt group is positioned horizontally.
	Sloped Square bolt group is sloped in the direction of the secondary part.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

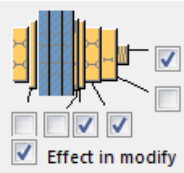


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

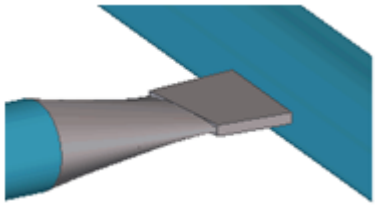
Squeezed tube (103)

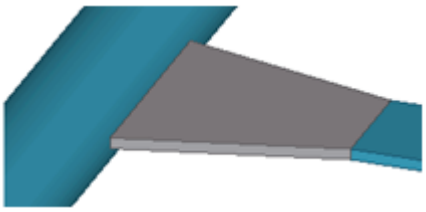
Squeezed tube (103) creates a squeezed part between two tube profiles. The squeezed part can be either a tube that is squeezed at one end and then welded to a plate, or a contour plate. It is also possible to define a connection where no squeezed tubes or contour plates are created, but the profiles are connected with cuts, fittings, and welds.

Objects created

- Squeezed tube and/or contour plate

Use for

Situation	Description
	Tube that is squeezed at one end and welded to a plate.
	Tube that is squeezed at one end and welded to a plate.

Situation	Description
	Contour plate.

Selection order

1. Select the main part.
2. Select the secondary part.

   The squeezed part is created.

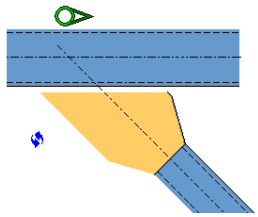
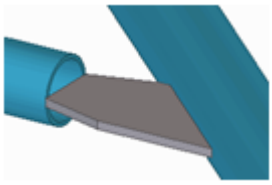
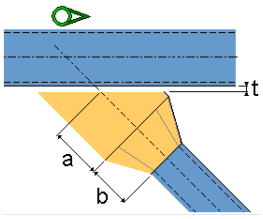
Picture tab

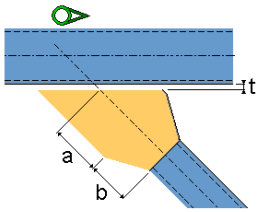
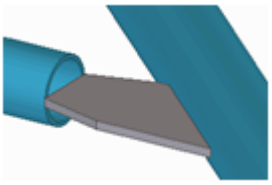
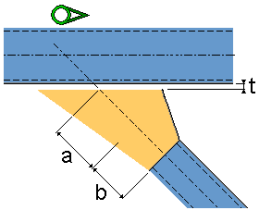
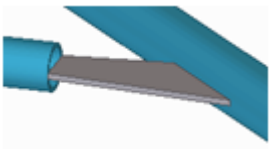
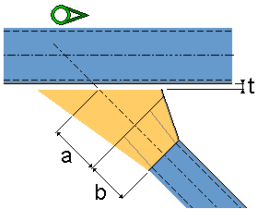
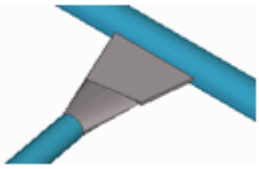
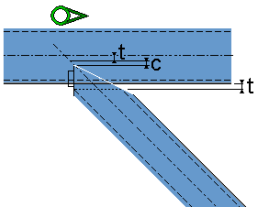
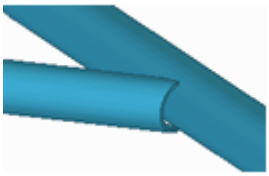
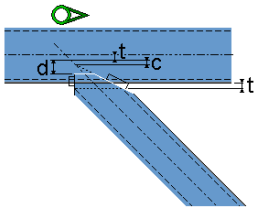
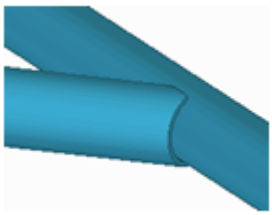
Use the **Picture** tab to define the shape and dimensions of the squeezed part, and whether the profiles are connected with a squeezed part or with cuts, fittings, or welds.

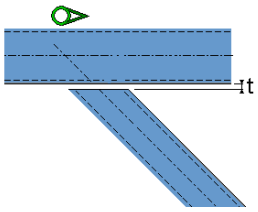
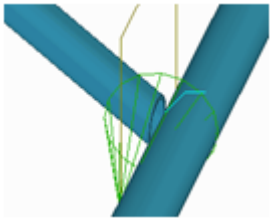
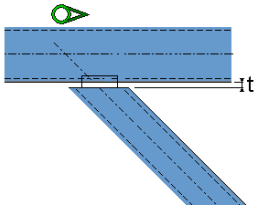
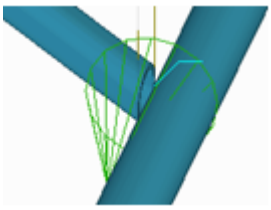
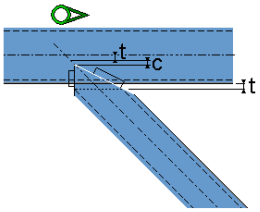
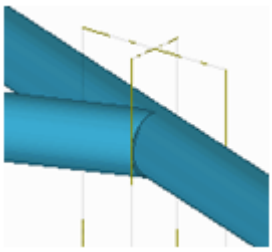
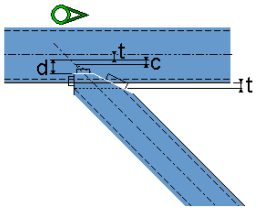
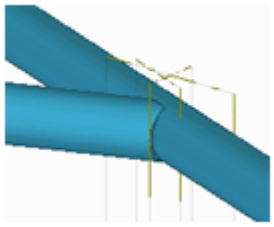
Squeezed part

The first five options create squeezed tubes and contour plates.

The last six options do not create new parts but line cuts, fittings and welds.

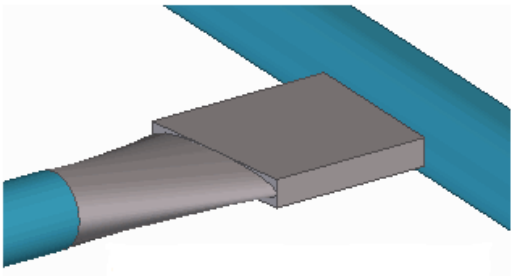
Option	Description	Example
	Default Main and secondary parts are tube profiles. A plate is used as a connecting profile, with one wide end and one parallel end.	
	A squeezed part and a contour plate are created. The tube profile reduces to a contour plate. You can define the length of the squeezed part using dimension **b**. Use dimension **t** to define the gap between the plate and the main part.	

Option	Description	Example
	Same as the **Default** option. Use dimensions **a** and **b** to define the length of the wide part and the parallel part of the plate. Use dimension **t** to define the gap between the plate and the main part.	
	Use dimensions **a** and **b** to define the plate length. The plate widens along its length. Use dimension **t** to define the gap between the plate and the main part.	
	Use dimensions **a** and **b** to define the plate length. The widening starts at the squeezed part and continues along the contour plate. Use dimension **t** to define the gap between the plate and the main part.	
	No new parts are created. The secondary part is adapted to the main part using a fitting and a line cut. Use dimension **c** to define an offset from the centerline of the main part (default value = `15 mm`). Use dimension **t** to define the gap between the plate and the main part.	
	No new parts are created. The secondary part is adapted to the main part using a fitting and two line cuts. Use dimension **d** to define the distance to the center of the main part. Use dimension **t** to define the gap between the plate and the main part.	

Option	Description	Example
	No new parts are created. The secondary part is shortened with a fitting. Optionally, the secondary part can be welded to the main part. Use dimension **t** to define the gap between the plate and the main part.	
	No new parts are created. The secondary part is shortened with a line cut. Optionally, the secondary part can be welded to the main part. Use dimension **t** to define the gap between the plate and the main part.	
	No new parts are created. The secondary part is shortened with a line cut. Optionally, the secondary part can be welded to the main part. Use dimension **t** to define the gap between the plate and the main part.	
	No new parts are created. The secondary part is shortened with a line cut. Optionally, the secondary part can be welded to main part. Use dimension **t** to define the gap between the plate and the main part.	

Parts tab

Use the **Parts** tab to define the thickness and width of the squeezed part end.

Option	Description
Squeezed part	Thickness and width of the squeezed part end. The front end of the squeezed part has the same profile as the secondary part. The squeezed part end and the contour plate are of the same size. 

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Comment	Add a comment about the part.	

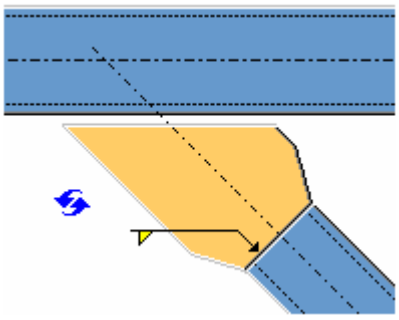
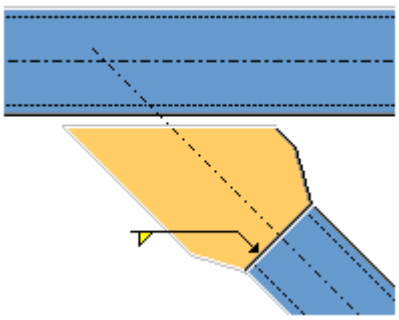
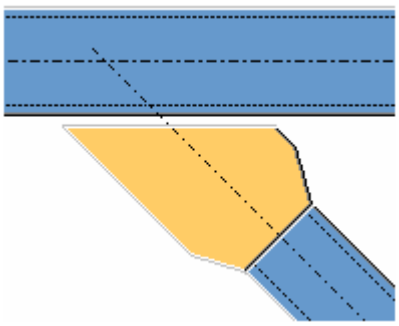
Parameters tab

Use the **Parameters** tab to define whether the secondary part is welded to the main part, whether the secondary part, the squeezed part and the contour plate are handled as a single part, and how the squeezed part is flattened.

Option	Description
B squeezed part	Define how the squeezed part is flattened. The options are: • **((De – t) * pi + t) / 2** (default) • **(De * pi) / 2** **De** = secondary tube profile

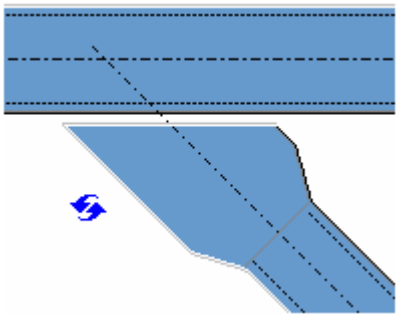
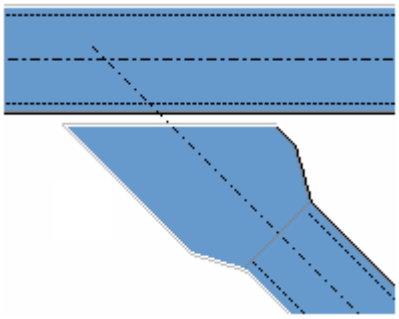
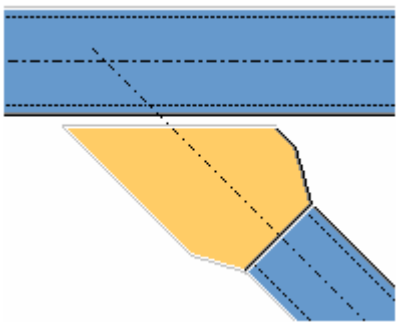
Welding

Define whether the secondary part is welded to the main part.

Option	Description
	Default Secondary part is welded to the main part. AutoDefaults can change this option.
	Secondary part is welded to the main part.
	Secondary part is not welded to the main part.

Part add

Define whether the secondary part, the squeezed part and the contour plate are handled as a single part.

Option	Description
	Default Secondary part, the squeezed part and the contour plate are not handled as a single part. AutoDefaults can change this option.
	Secondary part, the squeezed part and the contour plate are not handled as a single part.
	Secondary part, the squeezed part and the contour plate are handled as a single part. The squeezed part gets the profile properties from the secondary part.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

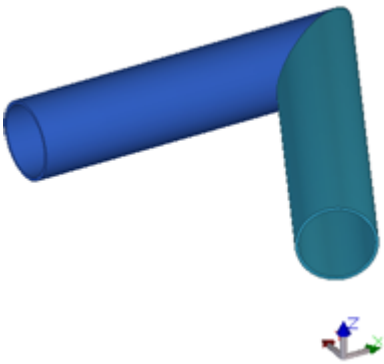
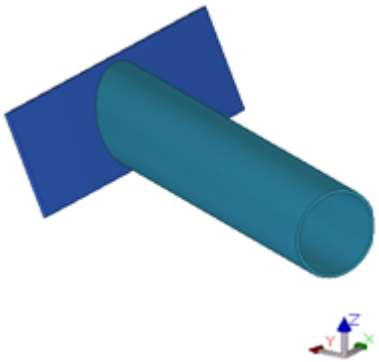
Tube-Chamfer

Tube-Chamfer connects a round tube to a plate or to a round tube. If the main part is tube, the secondary tube needs to be of equal diameter. The tubes are chamfered.

Objects created

- Cuts
- Welds

Use for

Situation	Description
	Tube-to-tube connection.
	Tube-to-plate connection.

Limitations

- No support for polybeams or contour plates.
- No support for non-plate profiles on the main part (for example, H/I beam or square tube)

Selection order

1. Select the main part (round tube or plate).

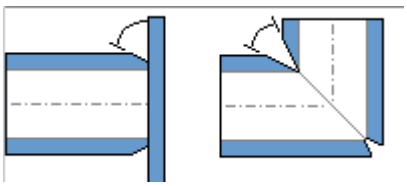
2. Select the secondary part (round tube of equal diameter).

   The connection is created automatically when the secondary part is selected.

Parameters tab

Use the **Parameters** tab to control the chamfers.

Connection options

Option	Description
	**Chamfer** Fits the secondary tube to a plate, or the main and secondary tubes along the median angle plane.
	Offshore Chamfer Creates back bevel during the tube NC file creation. No actual back bevel is created in the model.

NOTE **Offshore Chamfer** information is used only in NC data, and no actual back bevels are created in the model.

Tube NC Parameters

Option	Description
Bevel angle	The weld preparation angle created during NC processing of the tube. For a tube-to-tube chamfer this option creates a bevel to both tubes, adding up to the defined angle.
Root opening above	The gap between the parts. The gap is created in the model.
Shrinkage	The shrinkage considered during NC processing of the tube. The shrinkage value has no effect on the model.
Maximum torch angle	Maximum torch angle. The default is 70.0.
Minimum torch angle	Minimum torch angle. The default is -70.0.

NOTE **Bevel angle** information is used only in NC data, and no actual weld preparations are created in the model.

Welding tab

Click the link below to find out more:

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

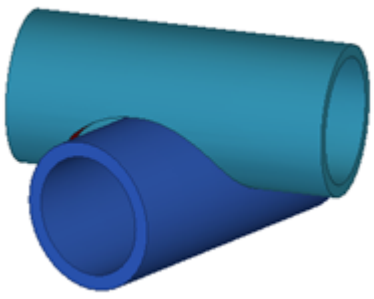
Tube-CrossingSaddle

Tube-CrossingSaddle connects a round tube to a round tube. The connection creates a saddle cut to the secondary tube.

Objects created

- Cuts
- Welds

Use for

Situation	Description
	Crossing saddle connection.

Limitations

- No support for polybeams or contour plates.

Selection order

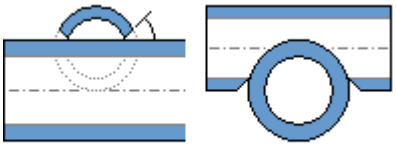
1. Select the main part (round tube).
2. Select the secondary part (round tube).

   The connection is created automatically when the secondary part is selected.

Parameters tab

Use the **Parameters** tab to control the saddle cuts.

Connection options

Option	Description
	**Crossing Saddle** Creates a crossing saddle to the secondary part.

Tube NC Parameters

Option	Description
Bevel angle	The weld preparation angle created during NC processing of the tube. The bevel is created to the secondary part.
Root opening above	The gap between the parts. The gap is created in the model.
Shrinkage	The shrinkage considered during NC processing of the tube. The shrinkage value has no effect on the model.
Maximum torch angle	Maximum torch angle. The default is 70.0.
Minimum torch angle	Minimum torch angle. The default is -70.0.

NOTE **Bevel angle** information is used only in NC data, and no actual weld preparations are created in the model.

Welding tab

Click the link below to find out more:

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

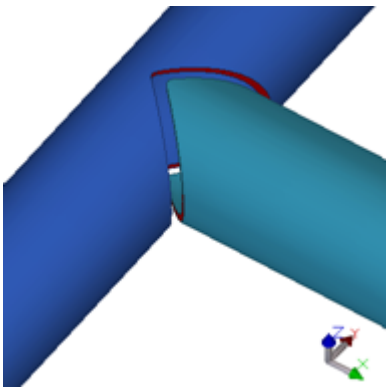
Tube-MitreSaddle+Hole

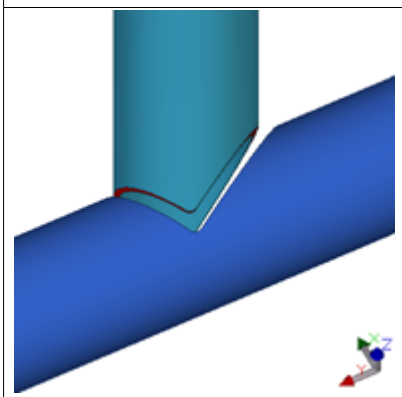
Tube-MitreSaddle+Hole connects a round tube to a round tube of equal diameter. The connection creates mitre holes to the main part and mitre cuts to the secondary part.

Objects created

- Cuts
- Welds

Use for

Situation	Description
	Mitre saddle and hole connection.

Situation	Description
	

Limitations

- No support for polybeams.
- Parts are only cut, not fitted. Short parts or parts at a steep angle may not show correct results.

Selection order

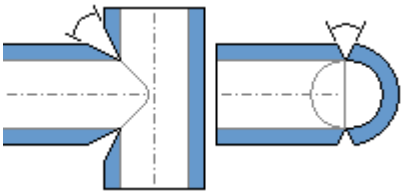
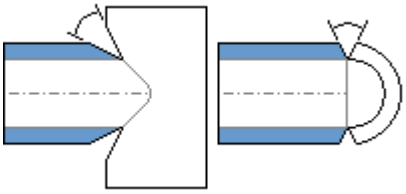
1. Select the main part (round tube).
2. Select the secondary part (round tube of equal diameter).

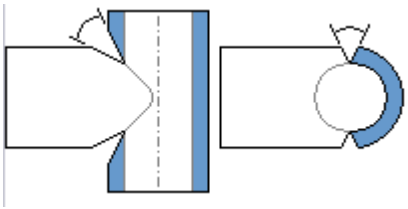
   The connection is created automatically when the secondary part is selected.

Parameters tab

Use the **Parameters** tab to control the mitre holes and cuts.

Connection options

Option	Description
	**Saddle + Hole** Creates a mitre cut (double mitre) to the secondary part, and a mitre hole to the main part.
	**Saddle Only** Creates a mitre cut to the secondary part. No mitre hole is created to the main part.

Option	Description
	**Hole Only** Creates a mitre hole to the main part. No mitre saddle is created to the secondary part.

Tube NC Parameters

Option	Description
Bevel angle	The weld preparation angle created during NC processing of the tube.
Root opening above	The gap between the parts. The gap is created in the model.
Shrinkage	The shrinkage considered during NC processing of the tube. The shrinkage value has no effect on the model.
Maximum torch angle	Maximum torch angle. The default is 70.0.
Minimum torch angle	Minimum torch angle. The default is -70.0.

NOTE **Bevel angle** information is used only in NC data, and no actual weld preparations are created in the model.

Weldings tab

Click the link below to find out more:

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

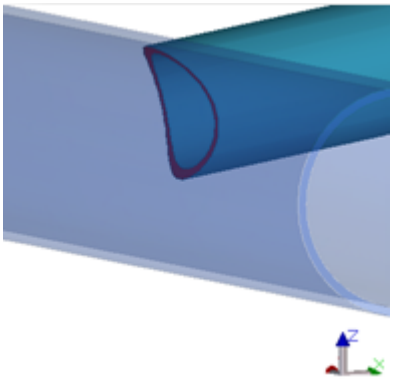
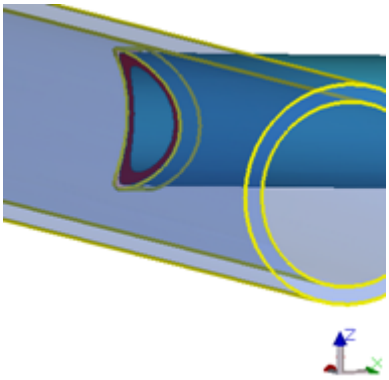
Tube-Saddle+Hole

Tube-Saddle+Hole connects a round tube to a round tube of equal or smaller diameter. The connection creates holes to the main part and saddles to the secondary part.

Objects created

- Cuts
- Welds

Use for

Situation	Description
 	Saddle connection.

Limitations

- No support for polybeams.
- The parts are only cut, not fitted.

Selection order

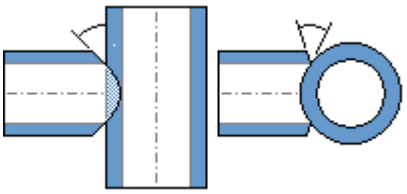
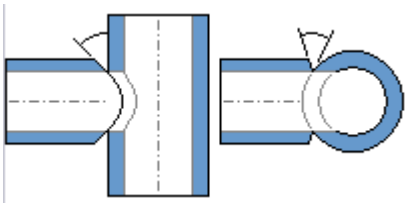
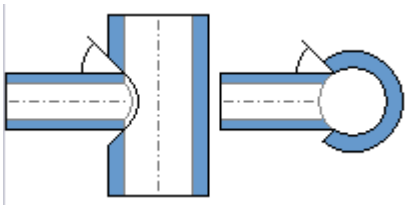
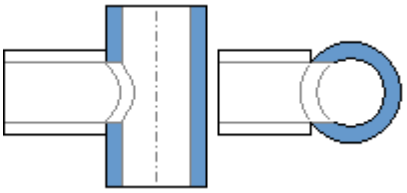
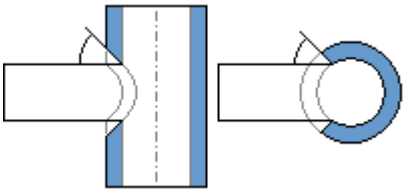
1. Select the main part (round tube).
2. Select the secondary part (round tube of equal or smaller diameter).

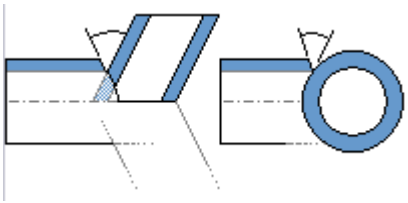
   The connection is created automatically when the secondary part is selected.

Parameters tab

Use the **Parameters** tab to control the saddles and holes.

Connection options

Option	Description
	**Saddle** Creates a standard saddle on the secondary part, fitting it to the main part. Weld preparations are created to the secondary part during NC processing.
	**Set-on Saddle+Hole** Creates a standard saddle on the secondary part, fitting it to the main part. Creates also a hole in the main part which matches the inner diameter of the secondary part. Weld preparations are created to the secondary part during NC processing.
	**Set-in Saddle+ Hole** Creates a saddle on the secondary part, fitting it to the inner surface of the main part. Creates also a hole in the main part which matches the outer diameter of the secondary part. Weld preparations are created to the hole on the main part during NC processing.
	**Set-on Hole Only** Creates a hole in the main part which matches the inner diameter of the secondary tube. The secondary part is not modified in any way. No weld preparations are created during NC processing.
	**Set-in Hole Only** Creates a hole in the main part which matches the outer diameter of the

Option	Description
	secondary part. The secondary part is not modified in any way. Weld preparations are created to the hole on the main part during NC processing.
	**Edge Saddle** Creates a partial saddle on the secondary part to a main part which only partially overlaps with the secondary part. Weld preparations are created to the secondary part during NC processing.
	Offshore Saddle Creates back bevel during the tube NC file creation. No actual back bevel is created in the model.

NOTE **Offshore Saddle** information is used only in NC data, and no actual back bevels are created in the model.

Tube NC Parameters

Option	Description
Bevel angle	The weld preparation angle created during NC processing of the tube.
Root opening above	The gap between the parts. The gap is created in the model.
Shrinkage	The shrinkage considered during NC processing of the tube. The shrinkage value has no effect on the model.
Maximum torch angle	Maximum torch angle. The default is 70.0.
Minimum torch angle	Minimum torch angle. The default is -70.0.

NOTE **Bevel angle** information is used only in NC data, and no actual weld preparations are created in the model.

Welding tab

Click the link below to find out more:

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

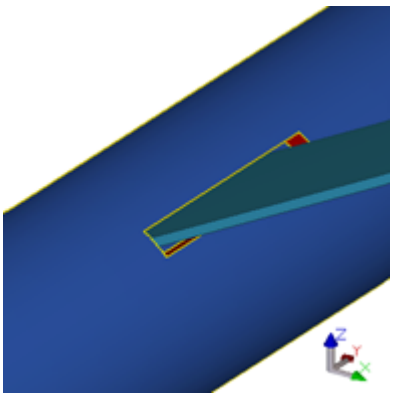
Tube-SlottedHole

Tube-SlottedHole connects a plate to a round tube. This connection creates a slotted hole to the main part.

Objects created

- Cuts
- Welds

Use for

Situation	Description
	Slotted hole with a plate.

Limitations

- No support for polybeams or contour plates.
- The hole is created only to one side of the main part. **Tube-SlottedHole** cannot be used for creating penetrating slots.

- Connections are not created for plates which are not parallel to the main axis of the tube.

Selection order

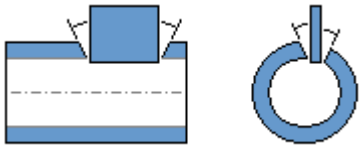
1. Select the main part (round tube).
2. Select the secondary part (plate).

   The connection is created automatically when the secondary part is selected.

Parameters tab

Use the **Parameters** tab to control the slotted holes.

Connection options

Option	Description
	**Slotted Hole** Creates a slotted hole to the main part.

Tube NC Parameters

Option	Description
Bevel angle	The weld preparation angle created during NC processing of the tube. The bevel is created to the slotted hole.
Root opening above	The gap between the plate and the tube. The gap is created in the model.
Maximum torch angle	Maximum torch angle. The default is 70.0.
Minimum torch angle	Minimum torch angle. The default is -70.0.

NOTE **Bevel angle** information is used only in NC data, and no actual weld preparations are created in the model.

Weldings tab

Click the link below to find out more:

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

5.11 Platework

This section introduces components that can be used in steel platework.

Click the links below to find out more:

- Rectangle to circle (17) (page 2052)
- Triangles generation (19) (page 2059)
- Unfold surface (21) (page 2068)

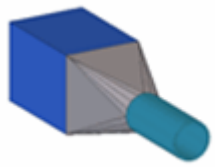
Rectangle to circle (17)

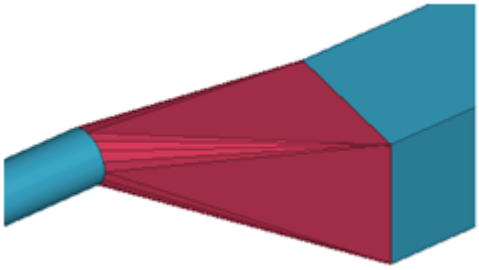
Rectangle to circle (17) creates a reducing piece between a rectangular profile and a circular or an elliptical profile. The reducing piece consists of several triangular plates.

Objects created

- Reducing piece

Use for

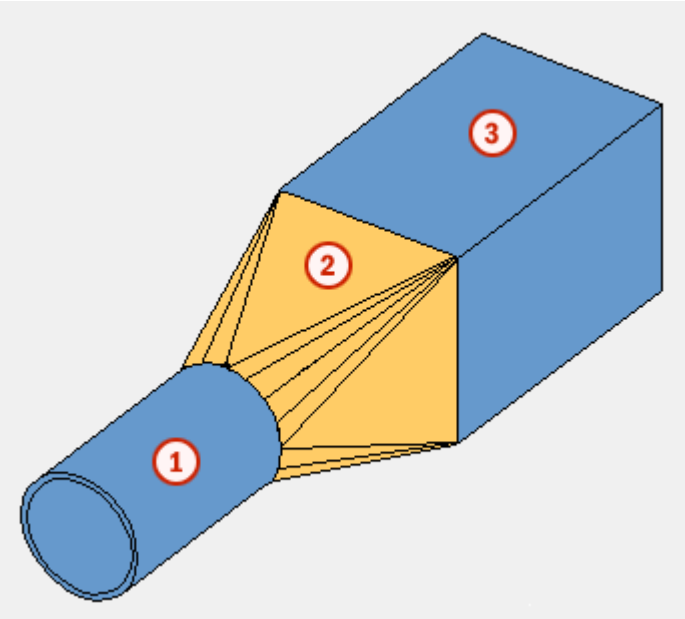
Situation	Description
	Reducing piece between rectangular and circular profile.

Situation	Description
	

Selection order

1. Select the main part (rectangular profile).
2. Select the secondary part (circular or elliptical profile).
3. Click the middle mouse button to create the reducing piece.

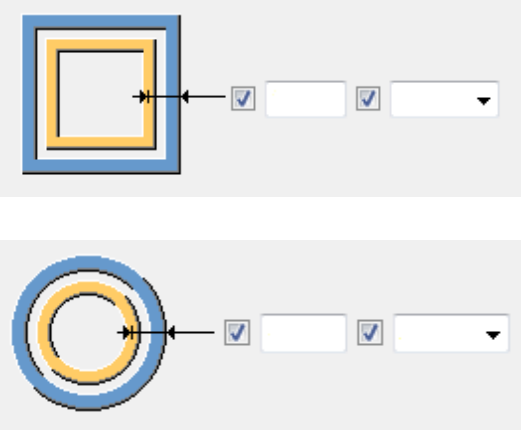
Part identification key



	Part
1	Circular profile
2	Reducing piece
3	Rectangular profile

Picture tab

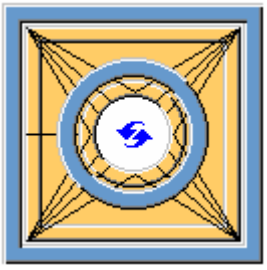
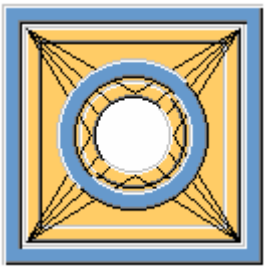
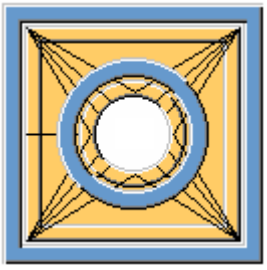
Use the **Picture** tab to define the number of parts which the reducing piece consists of, and the offset for the rectangular and circular profile end.

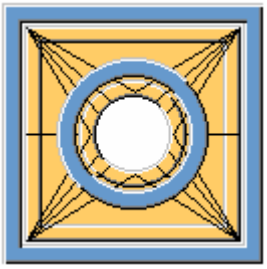
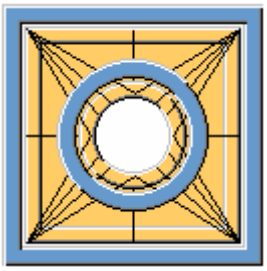
Option	Description
	Define the offset for the rectangular and the circular profile end. The options are: • **Offset** Fixed distance. • **% x t** Percentage of the plate thickness.

Number of cuts

Define the number of parts which the reducing piece consists of.

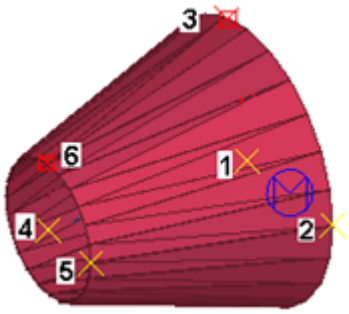
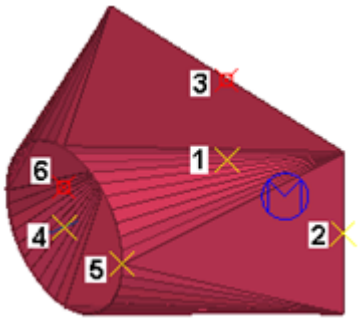
By default, there is one cut in the reducing piece.

Option	Description
	Default One cut AutoDefaults can change this option.
	No cuts
	One cut

Option	Description
	Two cuts
	Four cuts

Manual creation of reducing piece

You can create the reducing piece without existing profiles by picking three points at each end of the reducing piece. The picked points define the size of the reducing piece. You can define the shape of the manually created reducing piece on the **Parameters** tab.

Option	Description
	Picking order of the points: • center point • horizontal distance • vertical distance
	

Parts tab

Use the **Parts** tab to define the thickness of the triangular plates in the reducing piece and the reducing piece position.

Triangle

Option	Description
Triangle	Triangular plate thickness.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Comment	Add a comment about the part.	

Position in depth

Select the position of the plate segments. The default is **Middle**.

Parameters tab

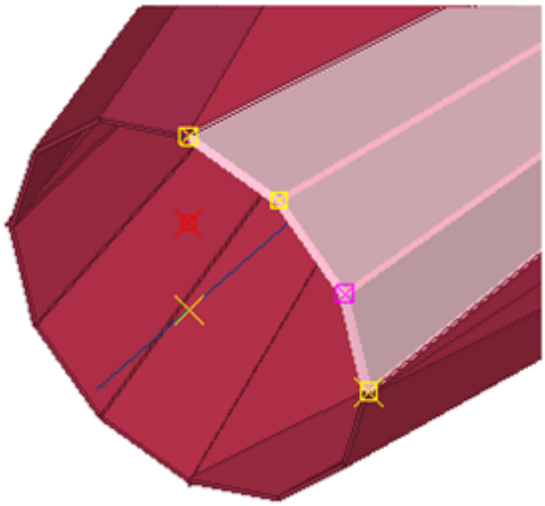
Use the **Parameters** tab to define the shape of the reducing piece end, to increase or decrease the number of triangular plates, and to define whether the triangular plates are welded.

Secondary segments

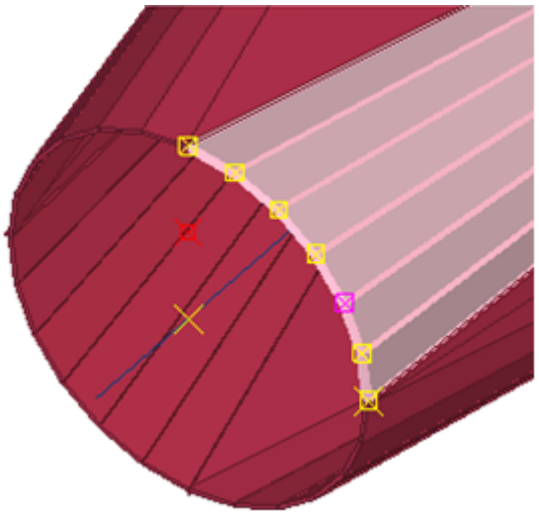
Define the number of the triangular plates in the reducing piece.

The more triangular plates are created, the more accurate is the shape of the reducing piece.

3 x 4 plates



6 x 4 plates

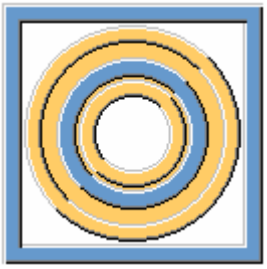
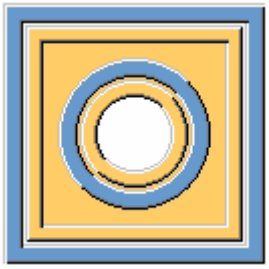
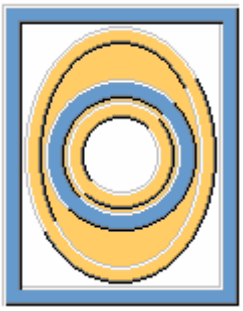
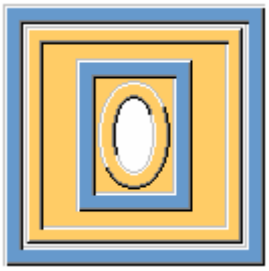
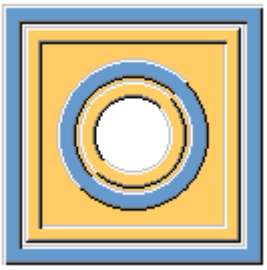
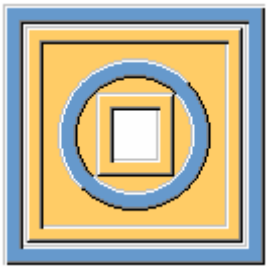


First/second shape

Define the shape of the reducing piece end if you have manually created the reducing piece by picking points in the order shown on the **Picture** tab.

By default, the shape of the reducing piece end is circular.

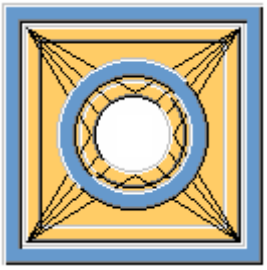
First shape	Second shape	Description
		Default Circle AutoDefaults can change this option.

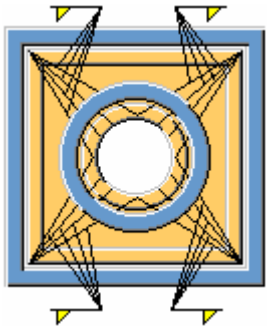
First shape	Second shape	Description
		Circle
		Ellipse This option works only if the main or secondary part has a rectangular profile.
		Rectangle

Welding of the plates

Define whether the triangular plates are welded.

Select the **Welding** option if you later need to show the unfolded assembly of triangular plates in an assembly drawing.

Option	Description
	Plates are not welded.

Option	Description
	Plates are welded.

Welds

Click the link below to find out more:

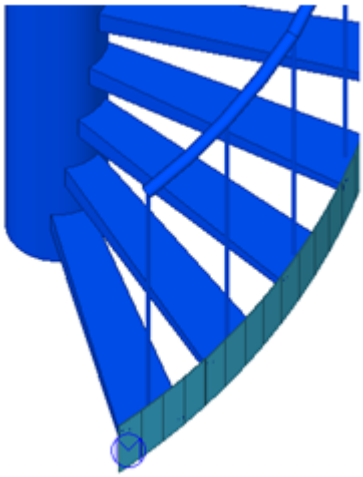
Triangles generation (19)

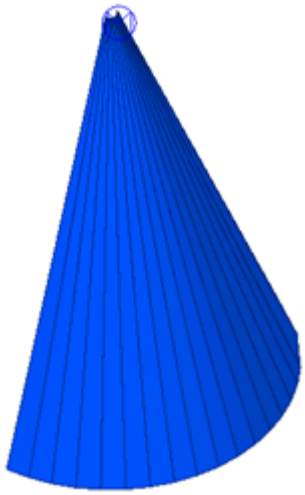
Triangles generation (19) creates triangular plates, or profiles, for double-curved surfaces, for example, for spiral stair stringers. A double-curved surface is created by placing multiple flat triangular plates side by side. The triangular plates are welded together to enable unfolding of the plates. If needed, you can unfold the triangular plates using **Unfold surface (21)**.

Object created

- Triangular plates

Use for

Situation	Description
	Curved surfaces consisting of triangular plates.

Situation	Description
	

Before you start

Create points in the model which define the shape of the curved surface. A minimum of 8 points are needed.

Alternatively, you can define the coordinates of the points in an ASCII file, and use the file to create the triangular plates. In ASCII files, the values are separated by spaces, and the decimals in the values are separated by periods, for example:

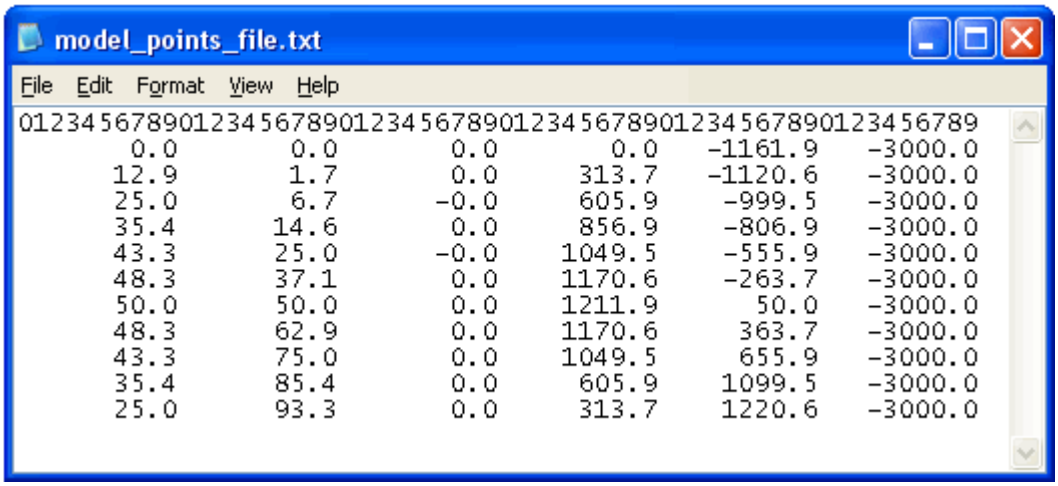
```
0.0 0.0 0.0 6000.0 0.0 -0.0
1620.7 -2010.1 500.0 6995.1 -3159.4 500.0
```

ASCII file example

The ASCII file for the coordinates of the points has a specific structure. A pair of coordinates is defined on each row with x-, y- and z-values.

The coordinates need to be placed at even distances from each other. Therefore, the first row contains a range of numbers that help to place the coordinates evenly.

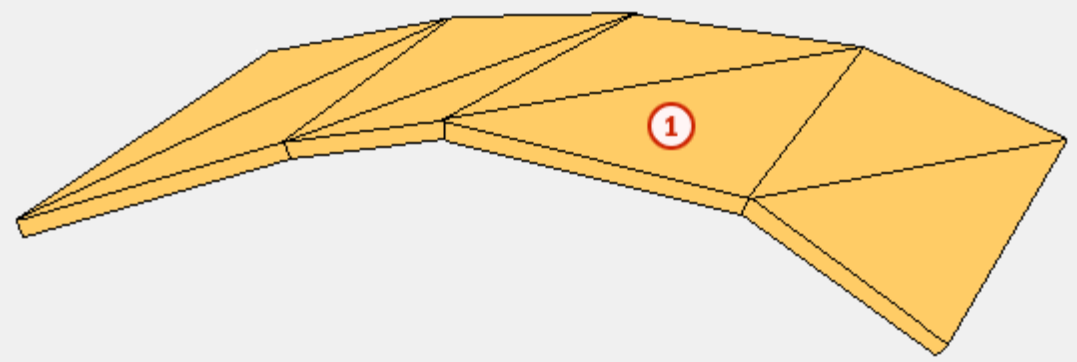
The coordinate pairs are defined on the next rows. The first three values define the local offset (x-,y-, z-) from the first point, and the last three values define the offset from the second point.



Selection order

1. Pick the points in the order shown on the **Picture** tab.
2. Click the middle mouse button to create the curved surface.

Part identification key



	Part
1	Triangular plate

Picture tab

Use the **Picture** tab to define whether the plates are created according to the picked points in the model or according to the coordinates defined in an ASCII file, and to set the global displacement.

Plate definition

Option	Description
With picked points	Shape of the triangular plate by picking the points that you have previously created.

Option	Description
Read points in ASCII file	Shape of the triangular plate by giving the coordinates in an ASCII file.

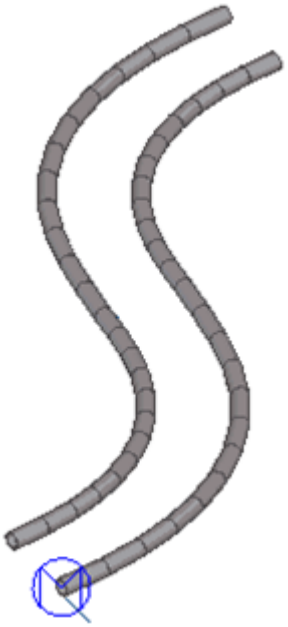
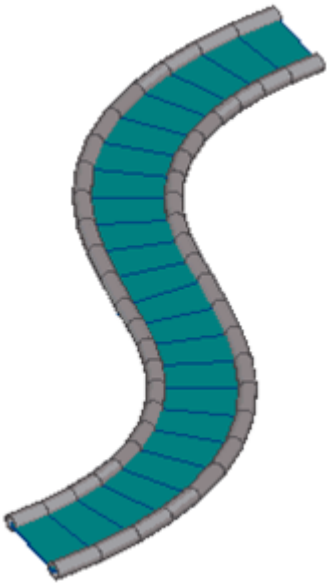
Offset

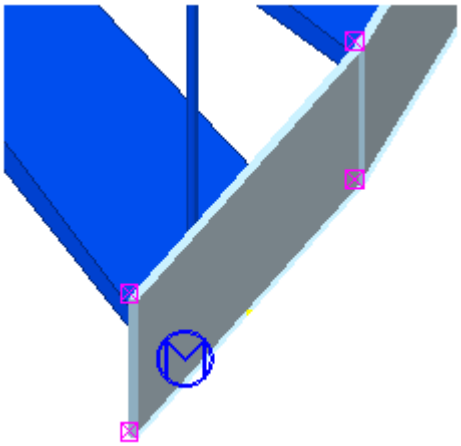
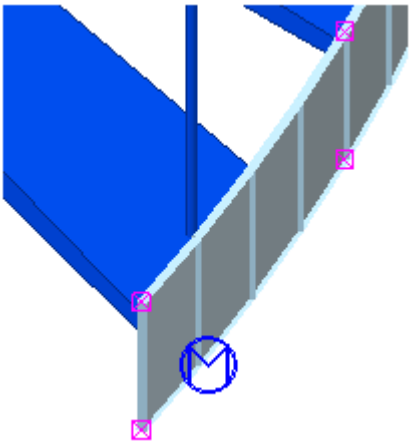
Use **Global displacement** to define an offset for the created plates or profiles in x, y and/or z direction.

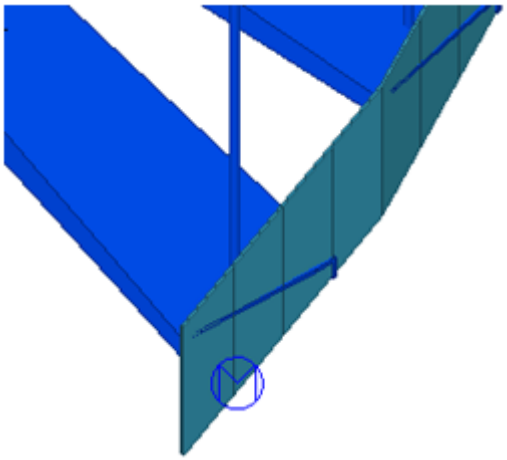
Parameters tab

Use the **Parameters** tab to define the ASCII file, whether plates or profiles are created, and how extra points are handled.

Option	Description
File name	Enter the name of the ASCII file where you have defined the coordinates for the points. The file is searched from the model folder.
Create plates profiles	Define whether plates or profiles are created. By default, plates are created. You can define the plate properties on the **Plate** tab and the profile properties on the **Profile** tab.
	Plates are created:

Option	Description
	Profiles are created: 
	Plates and profiles are created: 
Number of extra points	Define whether extra points are automatically created between the picked points, or set coordinates, to smoothen the plates.

Option	Description
	Number of extra points: 0 
	Number of extra points: 3 
Method to calculate extra points	If the curved plates form an arc, define whether the arc is taken into account for extra point calculation.

Option	Description
	The **1st order** option does not take the arc into account for extra point calculation. 
	The **3rd order** option takes the arc into account, and the extra points are placed in the same arc as the original points.
	The **Cardinal spline total length** option uses Cardinal spline interpolation over all points.
	The **Cardinal spline in middle** option uses Cardinal spline interpolation only for middle points.
Smoothness spline 0-1	Define the smoothing spline.
Coplane distance	Define the coplane distance.
Create points	Define whether points are placed on every coordinate.

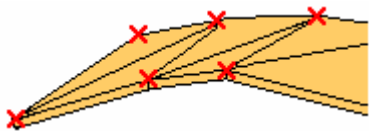
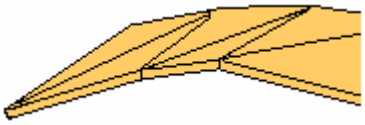
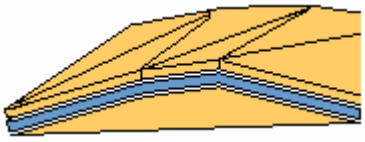
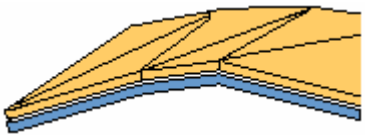
Option	Description
	Points are placed on every coordinate: 
	No points: 
Close curve	Define whether the contour is closed.
	Contour is closed: 
	Contour is not closed: 

Plate tab

Use the **Plate** tab to define the plate properties and position.

Option	Description
Plate	Triangular plate thickness.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Offset

Define the position of the triangular plates relative to the picked points or the coordinates.

Join plates

Define whether the triangular plates are attached to each other.

Select the **Welding** option if you later need to unfold the triangular plates. The triangular plates form an assembly which can be flattened by using **Unfold surface (21)**.

Profile tab

Use the **Profile** tab to define the profile properties and position.

Profile

Option	Description
Profile	Define the profile by selecting it from the profile catalog.

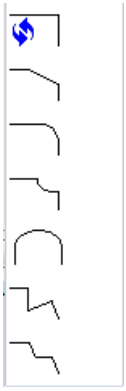
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Profile orientation

Use **Position in plane**, **Rotation** and **Position in depth** options to define the orientation of the profile.

Chamfers tab

Use the **Chamfers** tab to define chamfers for the created triangular plates.

	Select the chamfer shape.
X, **Y**	Enter the x and y direction coordinates.
Position	Select the relative position for each vertex. **Picked points** are the input points of the component. **Extra points** are points that the component creates. Enter the number of extra points on the **Parameters** tab to define the fragmentation of the result surface.
Angle conditions	Set the angle interval for each vertex. For example, if you set > 0 and < 90, all angles are between 0 and 90.

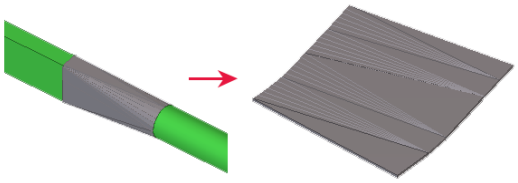
Unfold surface (21)

Unfold surface (21) unfolds welded plates. The unfolded plates are created to a location that you have defined. Use **Unfold surface (21)** to unfold triangular plates created with, for example, **Triangles generation (19)**. You can also create assembly drawings from the unfolded plates.

Objects created

- Unfolded plates

Use for

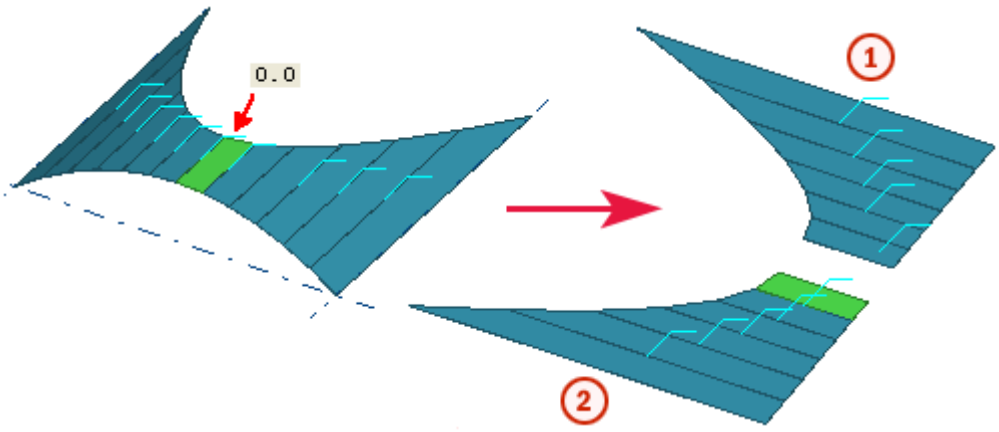
Situation	Description
	Unfolded triangular plates

Limitations

- **Unfold surface (21)** only works with contour plates. Do not use **Unfold surface (21)** to unfold beams or polybeams.
- We recommend that you use the same position-in-depth settings for all plates.
- In some cases, an error can occur when unfolding complex surfaces that have holes. You can use the **Do not refine edges** option on the **Big plate** tab to prevent this.
- **Unfold surface (21)** preserves anti-material cuts, line cuts, and bolts. Edge chamfers are not preserved.
- The plates should be neighbors with no gaps between them.
- The plates must be welded as neighbors. Welds must not have zero size.

NOTE If the weld size between the triangular plates is set to `0.0`, **Unfold surface (21)** unfolds only one triangular plate, not all the welded triangular plates.

If needed, you can create breaks in the unfolded shape using the weld sizes. For example, if the default weld size is `5.0`, but one weld gets size `0.0`, a break is created to the unfolded plate.



Option	Description
1	Unfolded form

Option	Description
2	Unfolded form

Selection order

1. Select a location for the unfolded plate.
2. Select a triangular plate.

   The unfolded plate is created to a location you have defined.

Plates tab

Use the **Plates** tab to define the thickness of the unfolded plate and whether the properties of the triangular plates are used in the unfolded plate.

Option	Description
Replace	Thickness of the unfolded plate. If you do not enter any value, the thickness of the triangular plates is used.

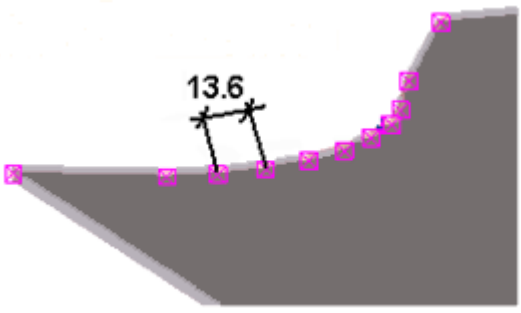
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

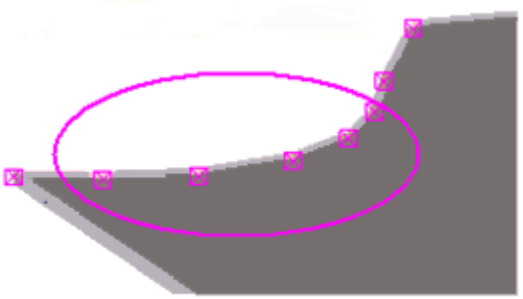
Option	Description
Zero welds	Select whether to ignore the plates that are connected by zero welds or not.
Unfolding by	Select how the plates should be unfolded: • By **Geometry** - in the order in which the original plates were created.

Option	Description
	• By **Welds** - in the way the pieces of original plates were welded together.
Profile	Select the second column of the check boxes if you want to use the properties of the triangular plates in the unfolded plate. If you are unfolding a plate created with **Rectangle to circle (17)**, and want to maintain the assembly position numbers defined in **Rectangle to circle (17)**, clear the second check box next to the **Profile** option.
Name	
Material	
Class	

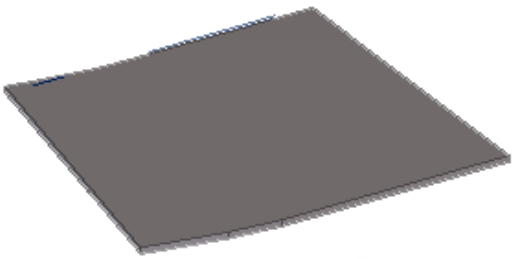
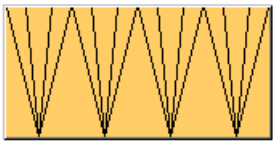
Big plate tab

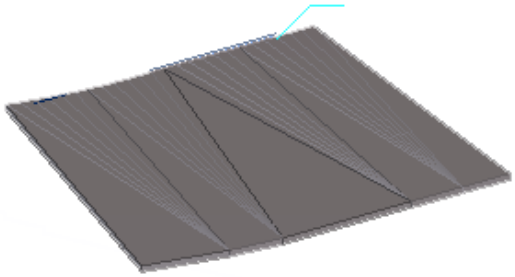
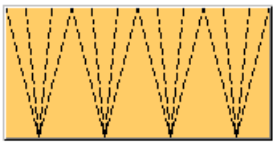
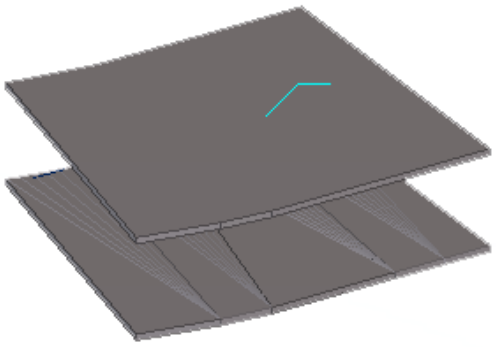
Use the **Big plate** tab to define the type and the accuracy of the unfolded plate.

Option	Description
Merge points	Define the accuracy of the unfolded plate by adding or removing points. Select whether the points in the resulting unfolded plate are merged or not.
Merge limit	If you want to merge points, define the limit for the merging. The points that are closer to each other than the set limit are merged. • Points are not merged. 

Option	Description
	• Points are merged, and the merge limit is `15.00`. 
Attach with	Select how the final plates are connected.

Unfolded plate type and offset

Option	Description
Type	**One plate** One plate is created. 
	Plates  Plates are welded together.

Option	Description
	
	Both  Both one plate and a plate with welds are created.  If you select **Both**, you can define the distance between the plates using the **Offset** option.

Offset

Define the distance between the unfolded plates.

Do not refine edges

This option prevents errors that may sometimes occur when unfolding complex surfaces that have holes.

Parameters tab

Use the **Parameters** tab to define the weld reference text in an assembly drawing.

Text in drawings

Option	Description
Prefix	First part of the text that is shown in assembly drawings, for example, `Angle=`.
Format	Format in which the size of the unfolding is displayed.
Postfix	Last part of the text that is shown in assembly drawings, for example, `degrees`.

UDA tab

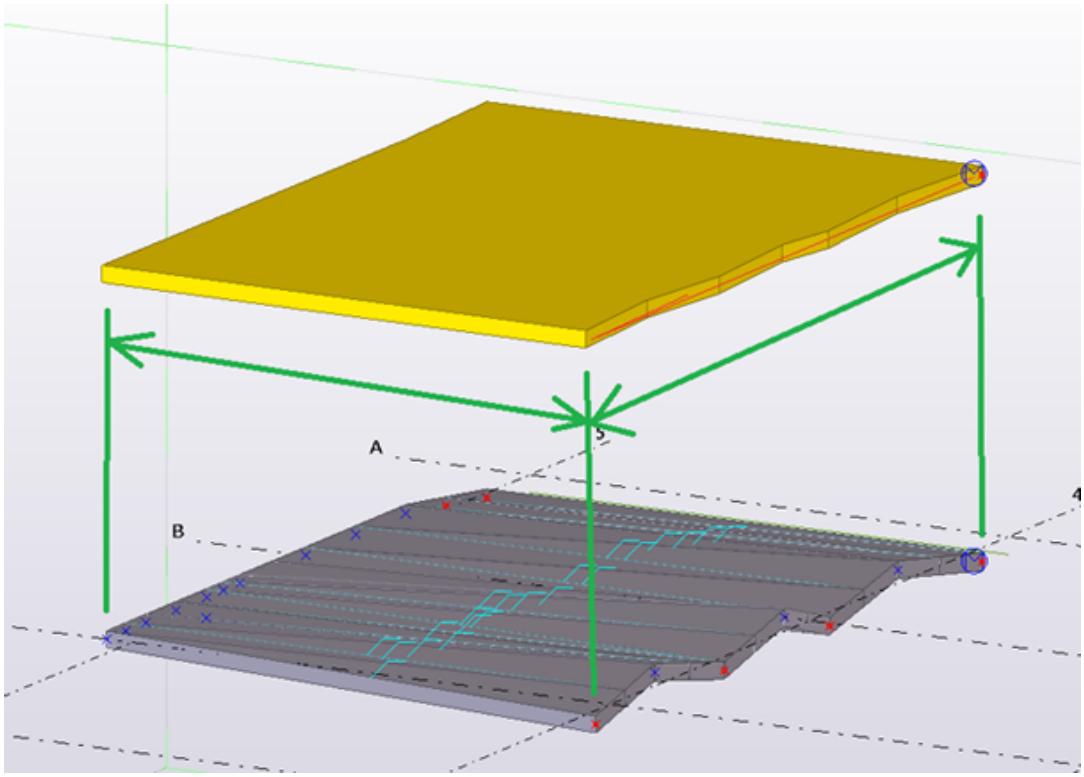
Use the **UDA** tab to copy the original numbering and user-defined attributes (UDAs) from the original plates to the unfolded plate.

Option	Description
Set following parameters	Copy the properties of the original plates to the UDAs of the unfolded plate. Enter the name of the UDA for each property that you want to copy.
Copy following UDAs	Define which UDAs are always copied from the original plates to the unfolded plate.

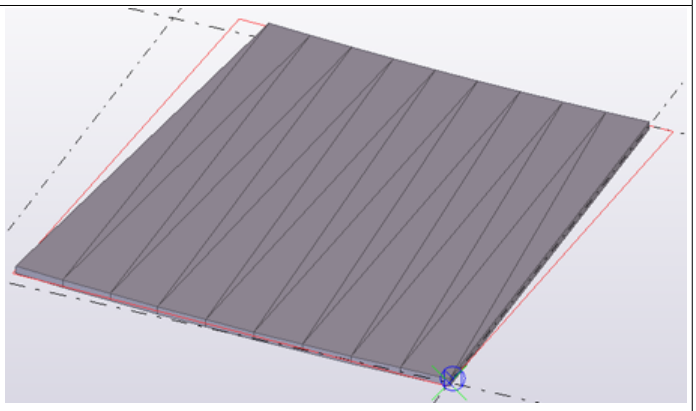
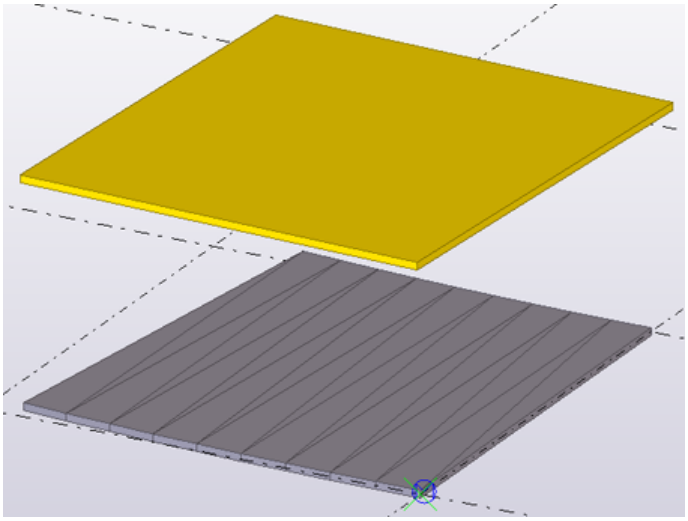
Surrounding rectangle tab

Use the **Surrounding rectangle** tab to calculate the length and width of the smallest rectangle around the unfolded area or the big plate.

The example image below shows the dimensions with green arrows.



Option	Description
Calculate surrounding rectangle	Select one of the following: • **No** (default). Surrounding rectangle is not created. You cannot enter any user-defined attribute (UDA) value or plate properties. • **Yes** Enter the UDAs for length and width. The rectangle is created with the dimensions defined in the UDAs. • **Yes and create construction lines** Enter the UDAs for length and width. The shape of the surrounding rectangle is shown with construction lines. Plates are unfolded by keeping the original smaller plates (mostly triangles), and with one big plate. Two rectangles are created: one around the original plates and one around the big plate.

Option	Description
	 • **Yes and create plate** Enter the UDAs for length and width, and the plate properties and the offset. The rectangle is created as a plate. The rectangular plate has the same thickness as the unfolded plate (defined on the **Plates** tab). Check on the **UDA** tab that you have entered the UDAs needed for the rectangular plate. 
UDA to put length **UDA to put width**	Enter the user-defined attributes (UDA) for length and width that are used in the surrounding rectangle. The calculated length and width cannot be saved if you do not enter the UDAs. Note that you have to enter both UDAs to save the calculated dimensions.

Option	Description
Rectangular plate	The rectangular plate has the same thickness, part position number, material, name and class as the unfolded plate. Define the UDAs for the plate on the **UDA** tab.
Offset	Define the perpendicular offset to the unfolded plate. By default, the offset is zero.

Rectangular plate properties

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

5.12 Frames

This section introduces components that can be used in steel frames.

Click the links below to find out more:

- Truss (S78) (page 2077)
- Opening Frame (page 2087)

Truss (S78)

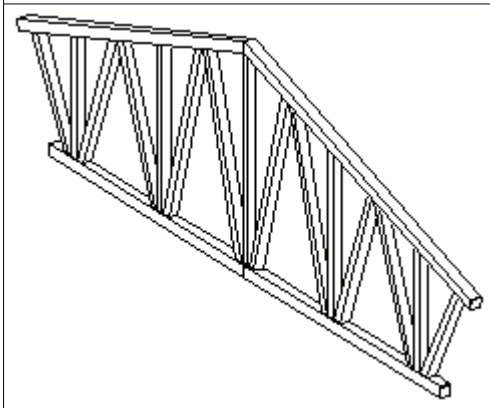
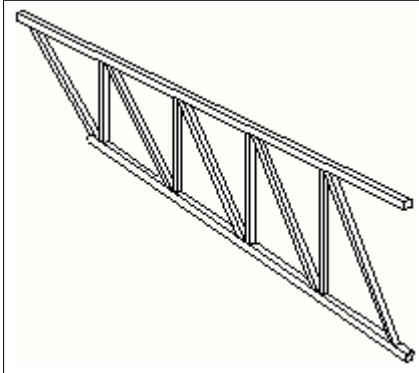
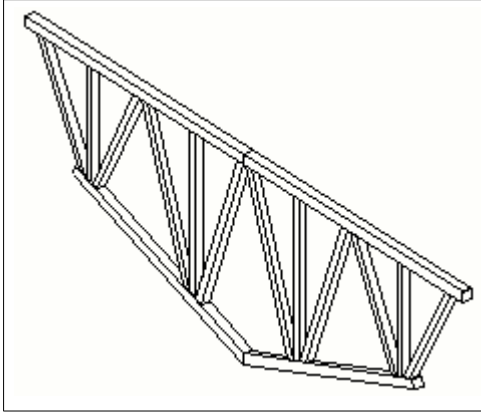
Truss (S78) creates a truss between selected points. **Truss (S78)** does not create any connections to existing parts.

Objects created

- Top chord

- Bottom chord
- Diagonals
- Verticals between diagonals
- Cap plates

Use for

Situation	Description
	Truss with top chord, tilted bottom chord, cap plates, diagonals and verticals.
	Truss with top chord, bottom chord, cap plates, diagonals and verticals.
	Truss with top chord, bottom chord, cap plates, diagonals and verticals.

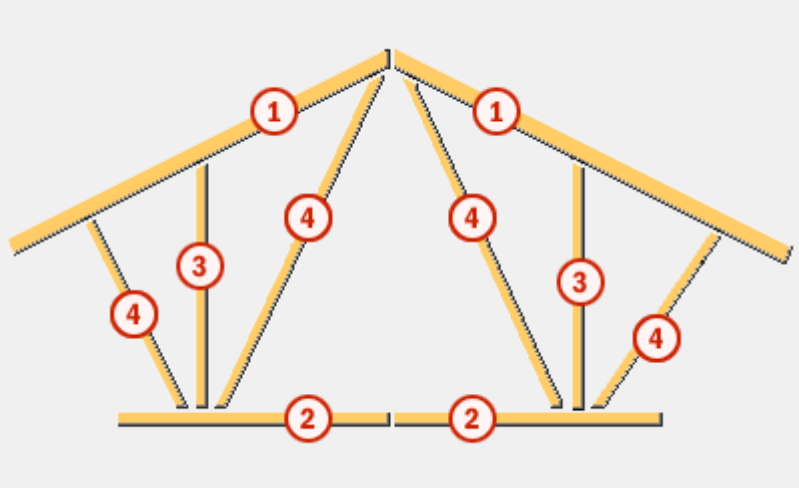
Selection order

1. Pick the start point of the truss.

2. Pick the end point of the truss.

   The truss is created automatically when the end point is picked.

Part identification key

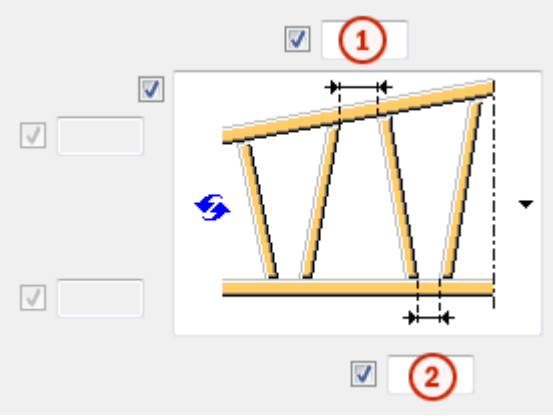


	Part
1	Top chord
2	Bottom chord
3	Vertical
4	Diagonal

Picture tab

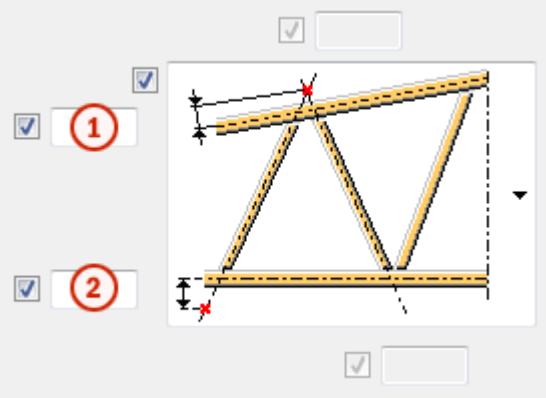
Use the **Picture** tab to control the gap created between diagonals, the eccentricity of the diagonals, and the dimensions of the parts.

Gap dimensions



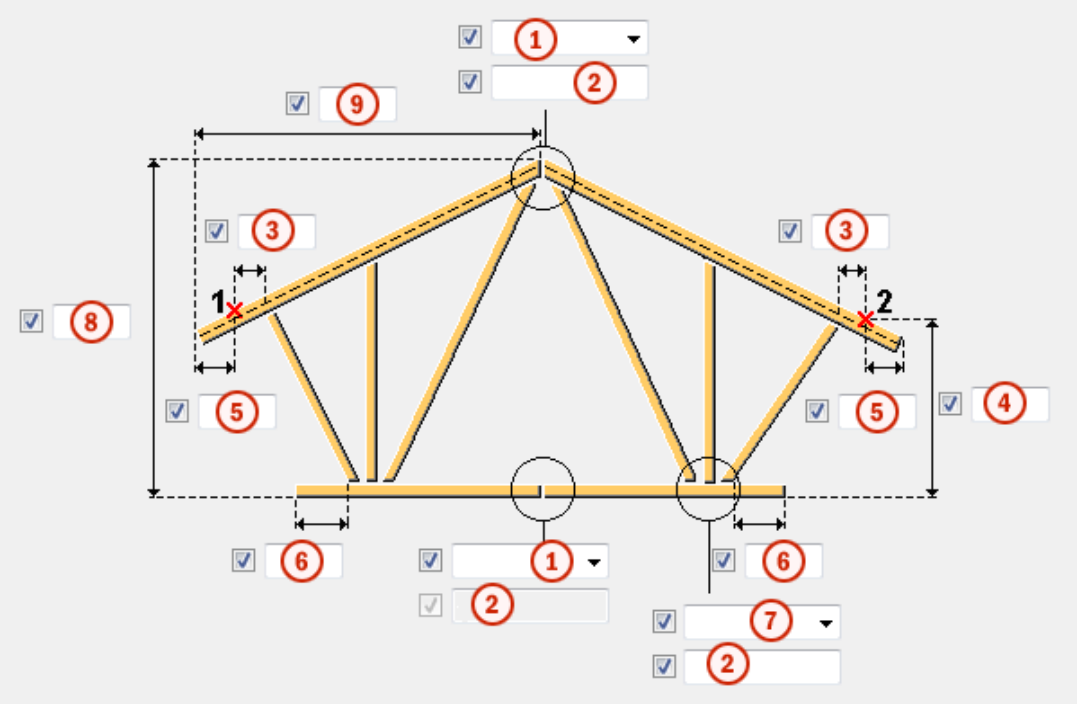
	Description	Default
1	Gap between the diagonals at the top chord.	20 mm
2	Gap between the diagonals at the bottom chord.	20 mm

Eccentricity dimensions



	Description	Default
1	Eccentricity of the diagonal intersection at the top chord.	20 mm
2	Eccentricity of the diagonal intersection at the bottom chord.	20 mm

Part dimensions



	Description	Default
1	Define how the top and bottom chords are connected. • **Apex haunch (106)** Use with top and bottom chord I profiles. • **Joining plates (14)** • **Welded** • **Continuous** Creates a continuous top or bottom chord.	`Welded`
2	Select an attribute file for the connection.	`standard`
3	Horizontal distance between the start/end point of the truss and the first/last diagonal.	`200 mm`
4	Vertical distance between the start/end point of the truss and the bottom level of the bottom chord.	`1000 mm`
5	Top chord extension from the start/end point of the truss.	`0 mm`
6	Bottom chord extension from the first and the last diagonal/vertical to the chord end.	`240 mm`

	Description	Default
7	Define how the chords, diagonals and verticals are connected. • **Gusset plate (11)** • **Round tube (23)** Used with round tube profiles. • **Welded**	`Welded`
8	Vertical distance between the truss apex and the bottom level of the bottom chord.	`2000 mm`
9	Horizontal distance between the top chord extension and the truss apex.	`entire truss length/2`

Parts tab

Use the **Parts** tab to control the profiles for chords, diagonals and verticals.

Profiles

You can define the profiles for the top and bottom chords, and up to seven profile types for the diagonals and verticals.

Use the **Diagonals** and the **Verticals** fields to define how the different profiles are created.

Option	Description	Default
Top Chord	Define the top chord profile by selecting it from the profile catalog.	`CFRHS100*4`
Bottom Chord	Define the bottom chord profile by selecting it from the profile catalog.	`CFRHS100*4`
Profile 1 - **Profile 7**	Define a profile by selecting it from the profile catalog.	`CFRHS80*4`

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Diagonals and verticals creation for left (1) and right (2) side

Options	Description
Diagonals	Define how the diagonals are created using the above profiles. • The diagonals are created by multiplying the number of diagonals with the type of profile, `number*profile`. For example, `2*3` creates two diagonals of the type **Profile 3**. For example, `1*2` is the same as `2`, and creates one diagonal of the type **Profile 2**. • The number of diagonals results from the pattern. For example, `2 2*3 1` creates 4 diagonals. • The diagonals are created from the start/end point towards the truss center. If you have set the **Truss type** to **Single pitch truss** on the **Parameters** tab, the second **Diagonals** row is ignored. The diagonals are created from the start point to the end point of the truss.
Verticals	Define how verticals are created using the above profiles. The verticals are positioned between the diagonals, and the maximum number of verticals results from the number of diagonals.

Twin profiles

Define whether the top or the bottom chord is created using twin profiles.

Top and bottom chord type

Option	Description
	Short legs up Default
	Short legs down
	Long legs up
	Long legs down

Diagonal and vertical type

Option	Description
	Short leg up Default
	Short leg down
	Long leg up
	Long leg down

Clearance

Define the gap between the twin profiles.

Parameters tab

Use the **Parameters** tab to control the truss assembly, and the diagonals and verticals creation.

Assembly main part

Define which part is the main part in the truss assembly.

- **Left Top Chord**

  Additional welds are created between the truss and the left top chord.
- **Right Top Chord**

  Additional welds are created between the truss and the right top chord.
- **Left Bottom Chord**

  Additional welds are created between the truss and the left bottom chord.
- **Right Bottom Chord**

  Additional welds are created between the truss and the right bottom chord.
- **None**

  The assembly main part is controlled by the bolts/welds that the **Truss (S78)** creates and the connections used between the parts. If all bolts/welds

are set to **Site**, then each part (top/bottom chords, verticals, diagonals) forms an individual assembly.

Verticals at truss end

Define whether a vertical is created to the truss ends.

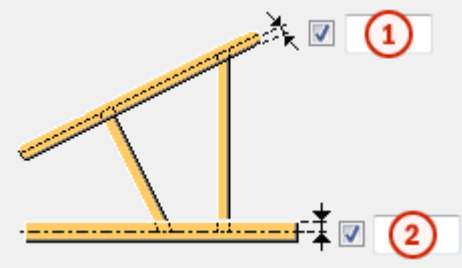
Option for start point	Option for end point	Description
		Default No vertical is created to the end. AutoDefaults can change this option.
		No vertical is created to the end.
		Vertical is created to the end.

Middle verticals

Define whether the verticals are created between diagonals.

Option	Description
	Default No middle verticals are created. AutoDefaults can change this option.
	No middle verticals are created.
	Verticals are created between the diagonals.
	Verticals are created between the diagonals. Verticals and diagonals form an N-type truss.

Extension for verticals and diagonals



	Description	Default
1	Extension for diagonals and verticals in the top chord.	0 mm
2	Extension for diagonals and verticals in the bottom chord.	0 mm

Truss type

Option	Description
	Default Single pitch truss AutoDefaults can change this option.
	Single pitch truss Top chord is horizontal. Bottom chord can be sloped.
	Ridge truss If the apex and the start/end point of the truss are of equal height, the chords are horizontal.
	Upside-down ridge truss If the apex and the start/end point of the truss are of equal height, the chords are horizontal.

Truss style

Option	Description
	Default WWW AutoDefaults can change this option.

Option	Description
	WWW The first diagonal begins from the top chord.
	AAA The first diagonal begins from the bottom chord.

Cap plate tab

Use the **Cap Plate** tab to control the cap plate creation.

Connection 1002 properties for cap plate

Cap plates are created using the **End plate detail (1002)** component that has a given set of properties. You can name the properties sets.

Top and bottom chord cap plates

Define whether a cap plate is created.

Option for left side cap plates	Option for right side cap plates	Description
		Default No cap plate is created. AutoDefaults can change this option.
		No cap plate is created.
		Cap plate is created.

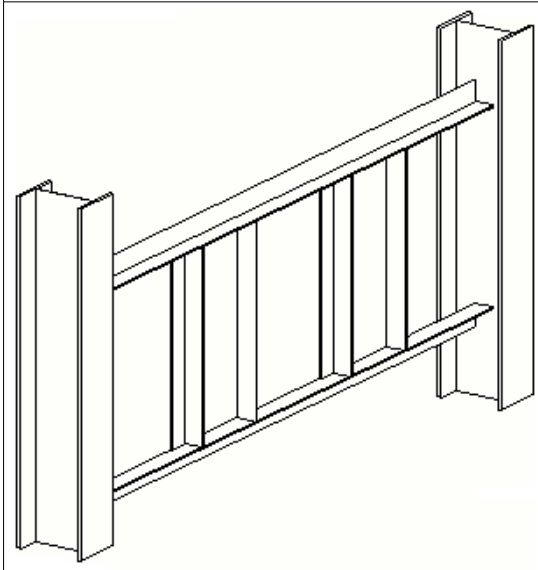
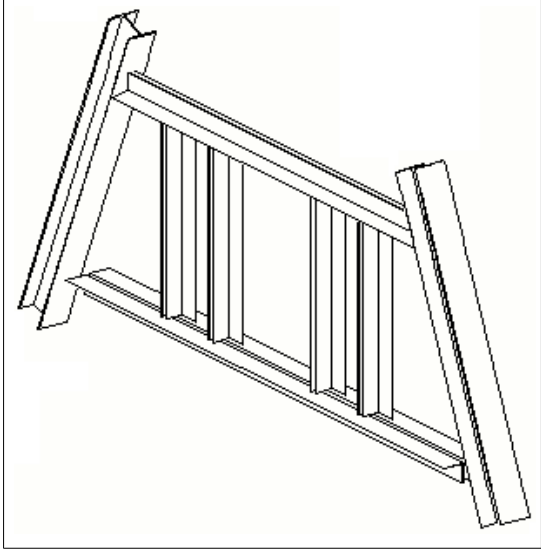
Opening Frame

Opening Frame creates an open frame for wall, roof, or floor openings. The frame is created between beams or columns.

Objects created

- Top frame
- Bottom frame
- Vertical posts
- Additional component (optional)

Use for

Situation	Description
	Open frame between two columns with top frame, bottom frame, and two pairs of vertical posts.
	Open frame between two skew columns with top frame, bottom frame, and two pairs of vertical posts.

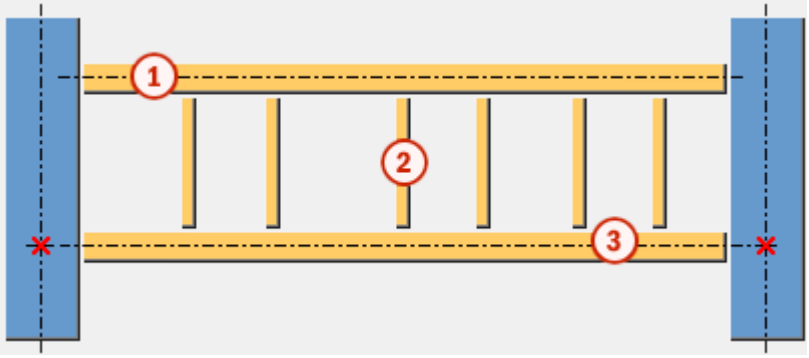
Before you start

Create two columns or beams.

Selection order

1. Select the main part.
2. Select the secondary part.
3. Pick the start point of the opening frame.
4. Pick the end point of the opening frame.

Part identification key

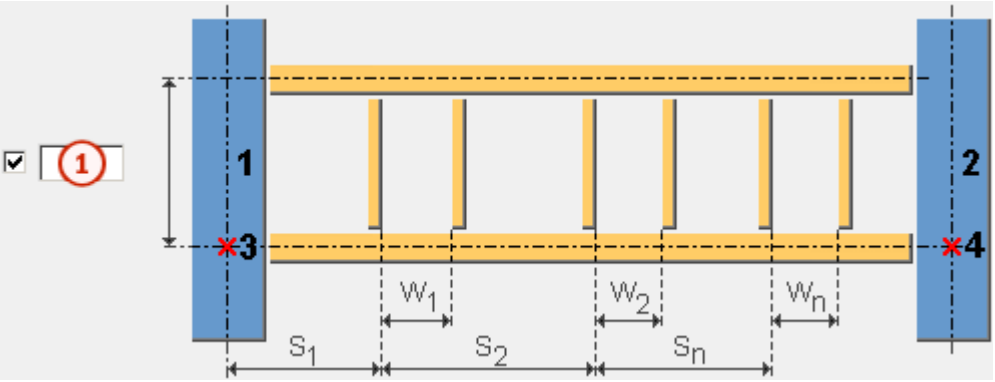


	Part
1	Top frame
2	Vertical post
3	Bottom frame

Picture tab

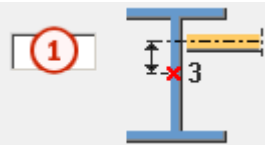
Use the **Picture** tab to control the frame position, offsets, and spacings.

Frame distance



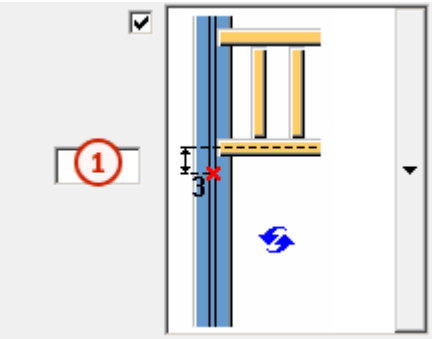
	Description	Default
1	Distance between the top and the bottom frames.	`1200 mm`

Frame horizontal offset



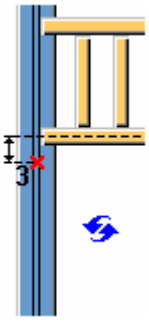
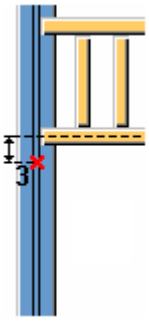
	Description	Default
1	Horizontal offset of the frame from the start/end point.	0 mm

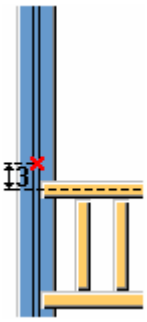
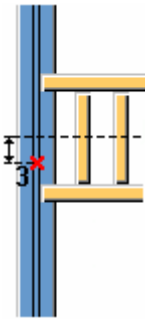
Frame vertical offset



	Description	Default
1	Vertical offset of the frame from the start/end point.	0 mm

Frame position

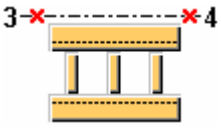
Option	Description
	Default Top AutoDefaults can change this option.
	Top

Option	Description
	Bottom
	Middle

Frame mirroring

Mirror the frame in relation to the start point and the end point.

When the frame is mirrored, also part rotation and additional connections follow the mirroring.

Option	Description
	Default Frame is not mirrored. AutoDefaults can change this option.
	Frame is not mirrored.
	Frame is mirrored.

Created parts

Option	Description	Default
Frames to create	Define whether top, bottom, or both frames are created.	`Both`
Pairs vertical posts	Define how many pairs of vertical posts are created.	`3`
Spacing pairs (S1, S2, Sn)	Spacing between the pairs. The spacing is measured from the first vertical post inner side of the first pair to the first vertical post inner side of the second pair. • If the spacing value between the posts exceeds the bottom frame length, then only top and bottom frames are created, and a warning message is displayed. • If the number of spacings is greater than the number of the entered spacing values, the missing spacing values are the same as the last spacing value. For example, if **Pairs vertical posts** = `4` and **Spacing pairs** = `100 200`, the spacing values are `100 200 200`.	`1800 mm`
Spacing vertical posts (W1, W2, Wn)	Spacing between the vertical posts in the pairs. The spacing is measured from the first vertical post inner side to the	`500 mm`

Option	Description	Default
	second vertical post inner side. If the number of spacings is greater than the number of the entered spacing values, the missing spacing values are the same as the last spacing value.	
Create welds	Define whether welds are created. The options are: • **Posts-Frames (5)** Welds are created only for posts on frames. • **Columns-Frames (1-4)** Welds are created only for frames on column. • **All** All welds are created. • **No** No welds are created.	`Posts-Frames(5)`

Parts tab

Use the **Parts** tab to define the part properties.

Dimensions

Option	Description	Default
Top Frame	Top frame profile by selecting it from the profile catalog.	`L100*50*5`
Bottom Frame	Bottom frame profile by selecting it from the profile catalog.	`L100*50*5`

Option	Description	Default
Vertical Posts	Vertical post profiles by selecting them from the profile catalog.	L100*50*5

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Twin profiles

Option	Description	Default
Twin profiles	• **No** Top frame, bottom frame, or vertical posts are created using single profiles. • **Yes** Top frame, bottom frame, or vertical posts are created using twin profiles. The twin profile is composed of the profile that you selected from the profile catalog.	No

Type

Define the profile rotation for single profiles or for twin profiles.

Options for single profile:

Option	Description
	Type 1
	Type 2
	Type 3
	Type 4
	Type 5
	Type 6
	Type 7
	Type 8

Options for twin profile:

Option	Description
	Type 1
	Type 2
	Type 3
	Type 4

Clearance

Option	Description	Default
Clearance	Clearance between the twin profiles. You can define the clearance only if the **Twin Profile** field is set to **Yes**.	`0 mm`

Position

Option	Description	Default
On plane	Part position on work plane.	`Middle`
Rotation	Define how much the part is rotated around its axis on the work plane. You can define the rotation only for twin profiles. Define the	`Front`

Option	Description	Default
	rotation for single profiles in the **Type** field.	
At depth	Part position, in terms of depth, perpendicular to the work plane.	`Middle`

Connections tab

Use the **Connections** tab to define the properties of connection components that are created between the parts.

Connection properties

NOTE With twin profiles the connection is created between only one part of the twin profile, and thus the connection is not well supported. If you use connections to connect twin profiles, a warning message is displayed.

Option	Description	Default
Connection number	Define a connection that connects the parts by selecting it from the component catalog. If the field is empty or set to `0`, welds are created instead.	
Attribute file	Select an attribute file for the connection.	`standard`

Welds tab

Click the link below to find out more:

5.13 Stairs

This section introduces components that can be used in steel stairs.

- Rail connection (70) (page 2097)
- Stairs (S71) (page 2110)
- Wooden steps pan (S72) (page 2130)
- Polybeam pan (S73) (page 2144)

- Handrail 1 (74) (page 2159)
- Z pan (S74) (page 2164)
- Kickplate (S75) (page 2199)
- Stanchions (S76) (page 2207)
- Railings (S77) (page 2214)
- Stairs (S82) (page 2246)
- Stanchion side plate (83) (page 2250)
- Multiple beam railing (S84) (page 2260)
- Stringer to channel (127) (page 2265)
- Stair base detail (1038) (page 2273)
- Stair base detail (1039) (page 2280)
- Stair base detail (1043) (page 2286)
- Ladder (S35) (page 2298)
- Cage ladder (S60) (page 2307)
- Ship Ladder (page 2321)
- Wall Rails (page 2341)

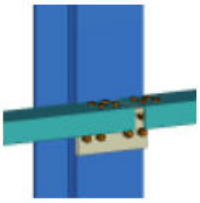
Rail connection (70)

Rail connection (70) connects an existing handrail and an existing column with a clip angle with either welded or bolted connections.

Objects created

- Clip angle
- Bolts
- Welds
- Cuts

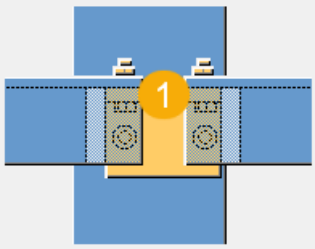
Use for

Situation	Description
	Handrail is connected to a column with bolted clip angle.

Selection order

1. Select the main part (column).
2. Select the secondary parts (handrail).
   - With continuous handrails, select the rail.
   - With spliced handrails, select the first rail and then the second rail.
3. Click the middle mouse button to create the connection.

Part identification key

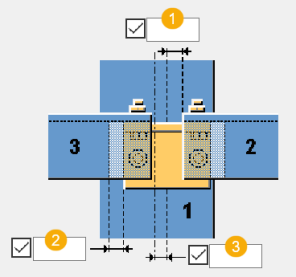


	Description
1	Clip angle

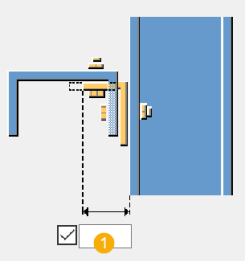
Picture tab

Use the **Picture** tab to define the connection dimensions.

Dimensions



	Description
1	Rail end cut back dimension.
2	Rail end clearance dimension from the clip angle.
3	Clip angle offset from the center line of the column.



	Description
1	Length of the outstanding leg of the clip angle.

Parts tab

Use the **Parts** tab to define the part properties.

Parts

Option	Description
Connecting profile	Select the profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Parameters tab

Use the **Parameters** tab to define the clip angle position and attachment type.

Clip angle location

Option	Description
	Default The clip angle is created below the handrail. AutoDefaults can change this option.
	The clip angle is created above the handrail.
	The clip angle is created below the handrail.

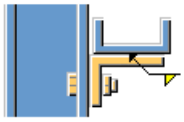
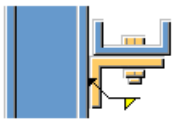
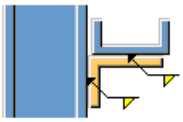
Clip angle direction

Option	Description
	Default The clip angle is created with the vertical leg toed down. AutoDefaults can change this option.
	The clip angle is created with the vertical leg toed down.
	The clip angle is created with the vertical leg toed up.

Outstanding leg position

	Default The long leg is positioned horizontally. AutoDefaults can change this option.
	Automatic Tekla Structures determines which way the long leg is positioned.
	The long leg is positioned horizontally.
	The long leg is positioned vertically.

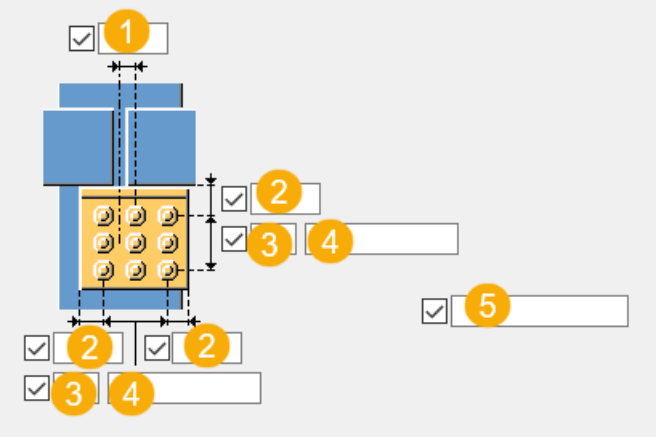
Attachment type

Option	Description
	Default The clip angle is bolted. AutoDefaults can change this option.
	The clip angle is bolted.
	The clip angle is welded to the handrail and bolted to the column.
	The clip angle is bolted to the handrail and welded to the column.
	The clip angle is welded.

Pbolts tab

Use the **Pbolts** tab to define the properties of the bolts connecting the clip angle to the main part.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.

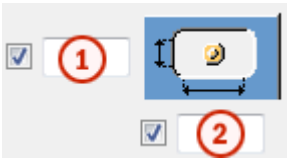
	Description
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

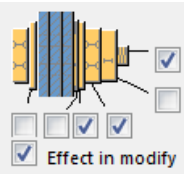


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

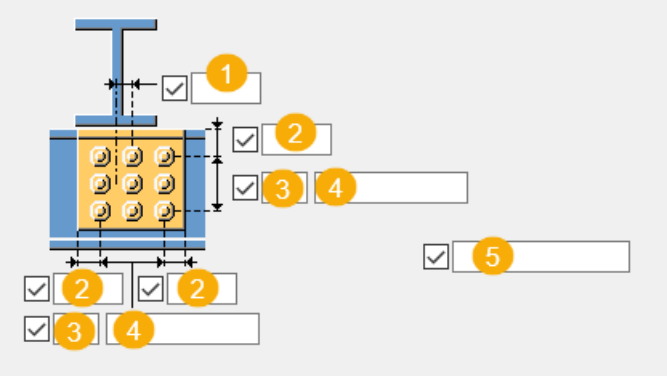
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Sbolts tab

Use the **Sbolts** tab to define the properties of the bolts connecting the clip angle to the secondary part.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

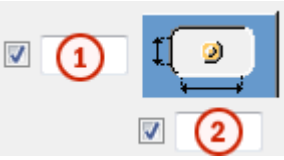
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

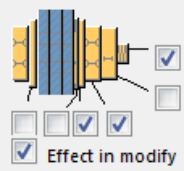


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

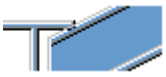
Automatic notching

Automatic notching options affect both the top and the bottom flange.

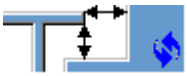
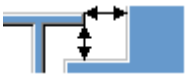
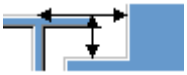
Notch shape

Automatic notching is switched on when you select a notch shape.

Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.

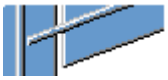
Option	Description
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

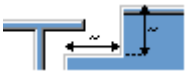
Enter the horizontal and vertical values for the cuts.

Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.

Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

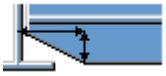
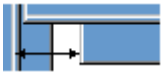
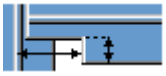
Side of flange notch

The side of flange notch defines on which side of the beam the notches are created.

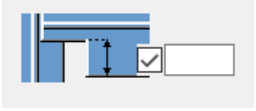
Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

Flange notch shape

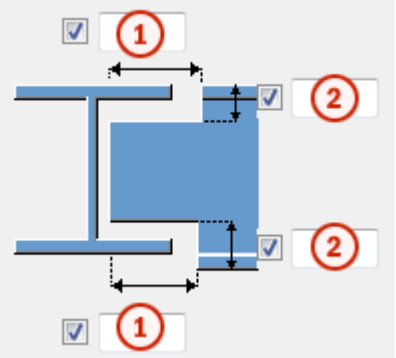
The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

Flange notch depth

Option	Description
	Flange notch depth

Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	10 mm
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest 5 mm.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

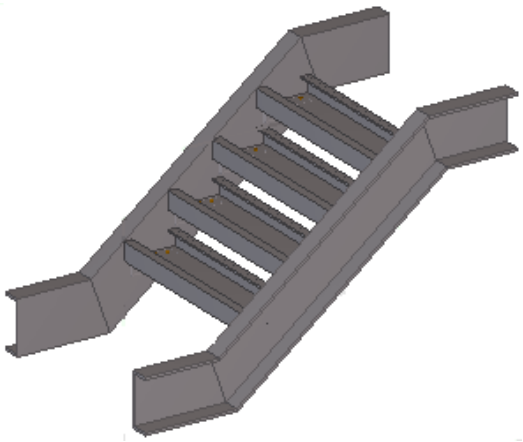
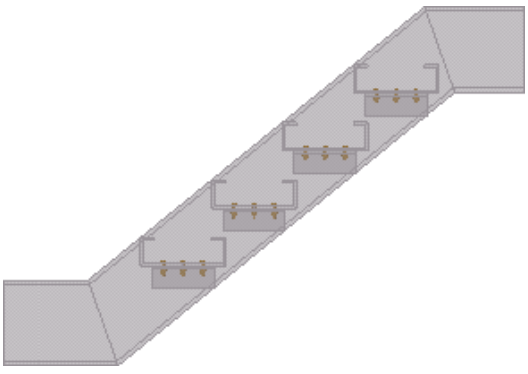
Stairs (S71)

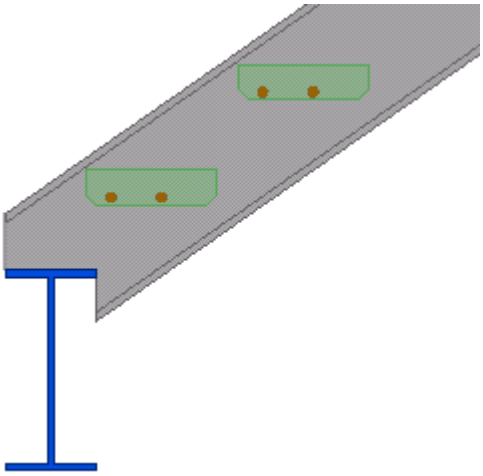
Stairs (S71) creates straight stairs with optional top and bottom landings. The stairs consist of stringers, possible landings and the actual steps.

Objects created

- Stringers
- Steps
- Landings (optional)
- Brackets
- Plates (optional)
- Cuts (optional)
- Bolts
- Welds

Use for

Situation	Description
	Stairs with horizontal top and bottom landings.
	U pan profile steps with brackets. Brackets are welded to the stringers and bolted to the steps.

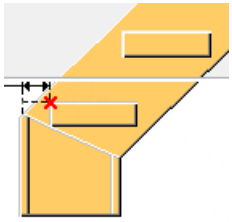
Situation	Description
	Stairs with stringers notched to the supporting beam. Steps are catalogue steps. You may also use your own custom components as steps.

Before you start

If the stringers are notched, create the supporting beams before creating the stairs.

Selection order

1. Pick a point to indicate the nosing point of the first step.

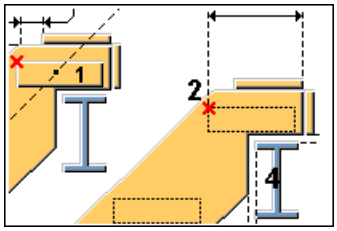


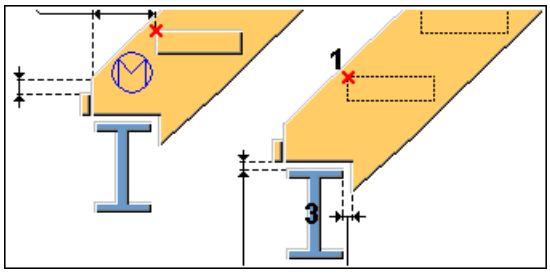
2. Pick another point to indicate the nosing point of the last step.

   Note that the order in which the points are picked has no effect.

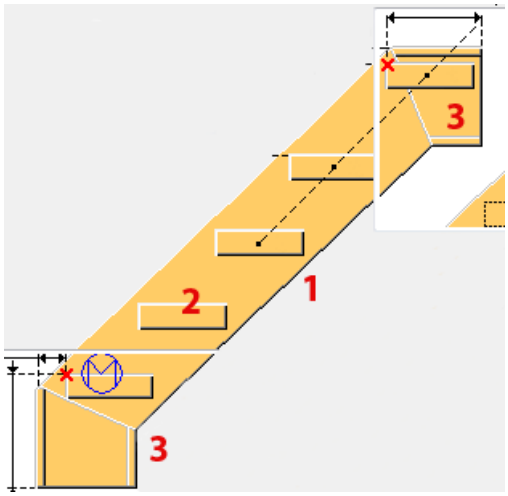
3. Click the middle mouse button to create the stairs.

   If you have selected the following notched options on the **Picture** tab, select the supporting beams and then click the middle mouse button.





Part identification key

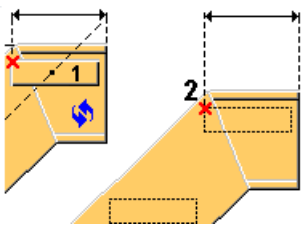


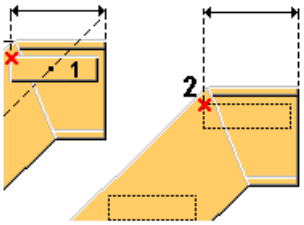
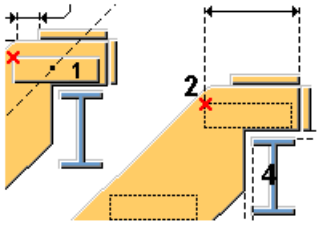
	Part
1	Stringer
2	Step
3	Landings

Picture tab

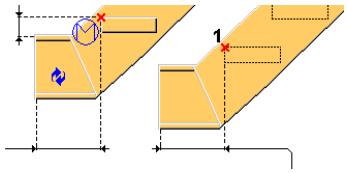
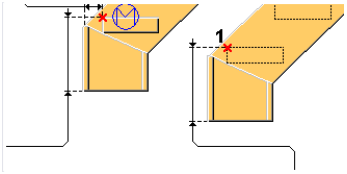
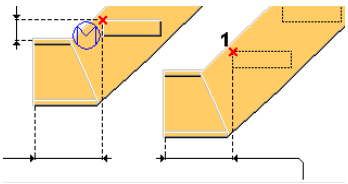
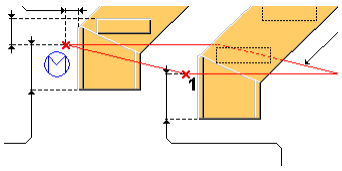
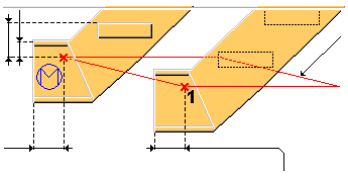
Use the **Picture** tab to control the landing types, and the shape and the location of the stringers in relation to the points you pick when creating the stairs.

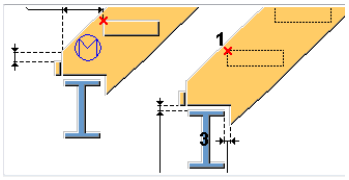
Top landing types

Option	Description
	Default Horizontal AutoDefaults can change this option.

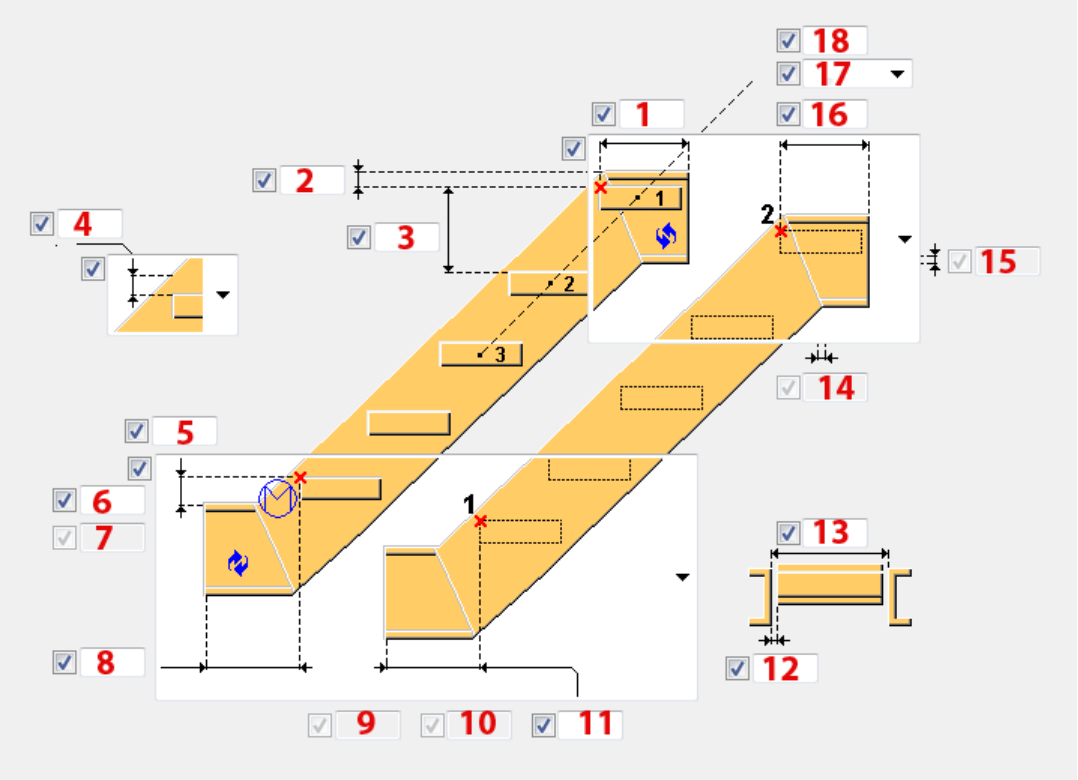
Option	Description
	Horizontal
	Notched The stringer top is on the supporting part. Vertical and horizontal plates are welded at the end of the stringer.

Bottom landing types

Option	Description
	Default Horizontal AutoDefaults can change this option.
	Vertical Creates a vertical landing where the lower point is on the top plane of the last step.
	Horizontal Creates a horizontal landing where the lower point is on the top plane of the last step.
	Vertical lowered point Creates a vertical landing where the lower point is at the top of steel or on the floor level.
	Horizontal lowered point Creates a horizontal landing where the lower point is at the top of steel or on the floor level.

Option	Description
	Notched Creates a notch to the bottom of the stringer. A vertical plate is welded at the end of the stringer.

Stair dimensions



	Description	Default
1	Define the distance between the upper point and the end of the left top landing. If you set the value to 0 mm, the left top landing is not created. If you set the top landing type to notched, define the cut dimension of the horizontal plate from the stringer edge.	0 mm

	Description	Default
2	Define the vertical distance from the upper point to the top of steel of the top landing. If the top landing is not created, the stringer top of steel is used. If you set the top landing type to notched, define the vertical distance from the upper point to the stringer top horizontal cut.	200 mm
3	Define the maximum distance allowed between two consecutive steps. The spacing between the steps is calculated using the step spacing type **Exact** or **Equal**, and the number of steps. For example, if the step height should be 200 mm, the **Exact** setting creates steps of exactly this height (enter the number of steps as an integer). The **Equal** setting creates the required number of steps from the bottom to the top, with the height calculated as close to 200 mm as possible.	(Z*220)/(Z+220) mm, where Z is the vertical distance between the 2 picked points.
4	Select the direction of the nosing dimension of the steps: • Vertical • Horizontal • Perpendicular Define the nosing dimension of the steps. This dimension depends on the direction you have selected.	Vertical 0 mm
5	Define this dimension based on the selected bottom landing type: • Horizontal or horizontal lowered point bottom landing. Define the vertical distance from the lower point to the top of steel of the bottom landing. • Vertical or vertical lowered point bottom landing. Define the horizontal distance from the lower picked point to the most distant face of the bottom landing.	150 mm

	Description	Default
	• Notched bottom landing. Define the horizontal distance from the lower point to the stringer vertical cut. By default, the cut is made at the supporting part edge.	
6	Define the vertical dimension from the lower point to the first step. You can define this dimension when the step spacing type is set to **Equal** and the bottom landing type is either vertical or horizontal lowered point.	Equal with vertical spacing between steps
7	Define the cut dimension of the vertical plate from the stringer edge. You can define this dimension when the bottom landing type is set to notched.	0 mm
8 **11**	Define this dimension based on the selected bottom landing type: • Horizontal or horizontal lowered point bottom landing. Define the horizontal edge distance between the lower point and the bottom landing. • Vertical or vertical lowered point bottom landing. Define the vertical edge distance between the lower point and the lowest point of the bottom landing.	600 mm
9	Define the vertical gap between the lower supporting part and the stringer cut. You can define this dimension when you have set the bottom landing type to notched.	0 mm
10	Define the horizontal gap between the lower supporting part and the stringer cut. You can define this dimension when you have set the bottom landing type to notched.	0 mm
12	Define the dimension to shorten the steps equally on both sides.	0 mm
13	Define the width of the steps.	1000 mm

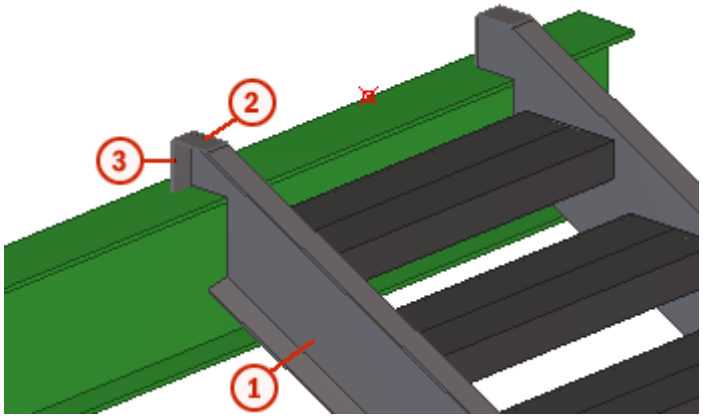
	Description	Default
14	Define the horizontal gap between the upper supporting part and the stringer cut. You can define this dimension when you have set the top landing type to notched.	0 mm
15	Define the vertical gap between the upper supporting part and the stringer cut. You can define this dimension when you have set the top landing type to notched.	0 mm
16	Define this dimension based on the selected top landing type: • Horizontal top landing. Define the distance between the upper point and the end of the stringer. • Notched top landing. Define the horizontal distance between the upper point and the stringer vertical cut. By default the cut is made at the supporting part edge. If you set this dimension to 0 mm, the top landing is not created.	0 mm
17	Select the step spacing type: • **Exact** creates the steps of exactly the defined height. Enter the number of steps as an integer. • **Equal** creates the required number of steps from the bottom to the top, with the height calculated as close as possible to the defined step height. The spacing type controls how the vertical spacing between the steps is calculated. If you select **Exact** and set the bottom landing type to horizontal or vertical lower point, the steps are spaced from the upper point to the lower point. In all other cases the steps are spaced from the lower point to the upper point.	**Equal**

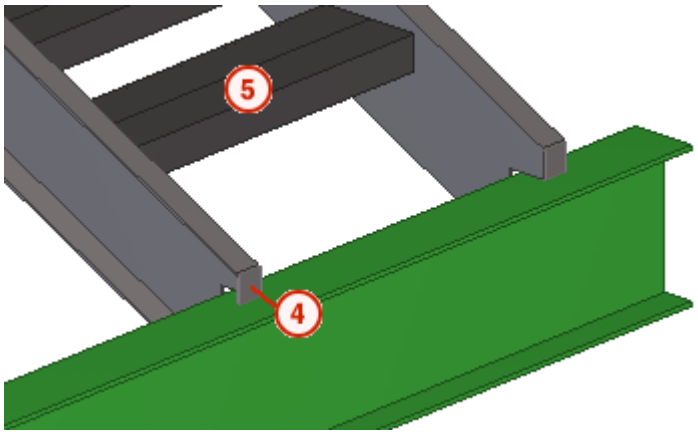
	Description	Default
18	Enter the number of steps. This option depends on step spacing type and step spacing. If you set the step spacing type to **Exact**, or to **Equal** and the spacing distance is not defined, **Stairs (S71)** creates the number of steps you have defined.	

Stair set up tab

Use the **Stair setup** tab to control the part properties, top and bottom steps, position of the stairs on the horizontal plane, and rotation of stringers and steps.

Stair part properties





	Part	Description	Default
1	**Left stringer**	Always created Select a profile from the profile catalog.	`BLU400*2` The default name is `STRINGER`.

	Part	Description	Default
1	**Right stringer**	Always created Select a profile from the profile catalog.	BLU400*2 The default name is STRINGER.
2	**Upper H plate**	Created only if the top of the stringer is on the supporting beam.	6 mm The default name is PLATE.
3	**Upper V plate**	Created only if the top of the stringer is on the supporting beam.	6 mm The default name is PLATE.
4	**Lower V plate**	Created only if the bottom of the stringer is on the supporting beam.	6 mm The default name is PLATE.
5	**Catalogue step**	Created only if **Step type** is set to **Catalogue step**. In this case, the U-pan steps, brackets, and bolts defined on the other tabs are ignored.	First profile from the **Catalogue step** list, defined in the steps.dat text file in the system (page 446) folder. The list changes according to the selected environment. The default name is STEP.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

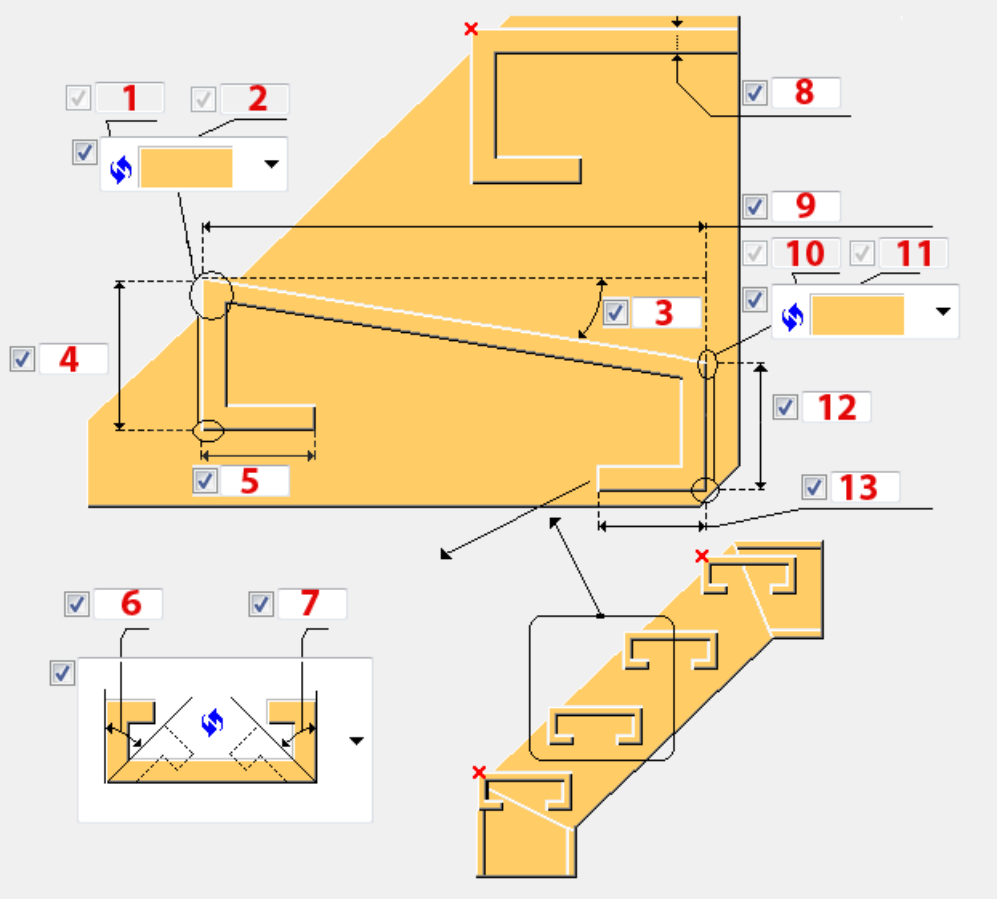
Stair setup settings

Option	Description
Step type	• **Steps**: The steps are created from the steel pan that is defined on the **Steps** tab. • **Catalogue step**: The steps are created from the profile defined in the **Catalogue step** option. The options on the **Steps** and **Bracket** tabs are not used.
Create assembly	Select which parts of the stair component form an assembly. The default is stringers.
Stringer reference line	• **Above**: The line positioned on the inside of the stairs on the top face is used as the reference line. • **Nosing**: The nosing line of the steps is used as the reference line. The reference line is used for workshop drawings.
Steps rotation	Rotation of the step around its axis. The default is **Top**.
Position in plane	Position of the stairs. The middle line of steps is used as the reference line. The default is **Right**.
Offset	Offset of the stairs on the plane from the position that is set in the **Position in plane** option. The default offset is `0 mm`.
Bolt Type	Select the bolt type for catalogue steps. If you select the **Workshop** option, the bolts are shown in workshop assembly drawings in the list of workshop bolts.
Bolt tolerance	Define the bolt hole tolerance for catalogue steps.
Stringer rotation	Rotation of the stringer around its axis. The default is top .
Create top step	Define whether the first step of the stairs (the highest step) is created. By default, the first step is created.
Create bottom step	Define whether the last step of the stairs (the lowest step) is created. By default, the last step is created.

Steps tab

Use the **Steps** tab to control the shape and the size of U pan steps. Instead of a U pan, you can create the steps by using a catalog profile, or a custom part or component.

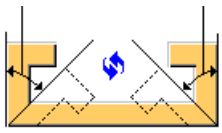
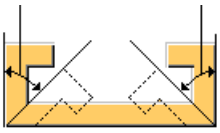
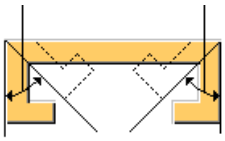
U-pan step dimensions



	Description	Default
1, **2**, **10**, **11**	Define the chamfer values on both sides of the step. Select the chamfer type to set the shape of the front and back corner of the step.	15 mm
3	Define the angle of the step relative to the horizontal line. The angle can be positive or negative.	0 degrees
4, **12**	Define the height of the step.	100 mm
5, **13**	Define the length of the horizontal portion of the U pan.	50 mm

	Description	Default
6, **7**	Define the angle of the vertical portion of the step relative to the vertical line. The angle can be positive or negative.	`0` degrees
8	Define the thickness of the U pan step material.	`10 mm`
9	Define the width of the step.	`Bracket length * 1/0.7`

Step type

Option	Description
	Default U pan legs up AutoDefaults can change this option.
	U pan legs up
	U pan legs down
	Custom Define the custom profile options.

Custom profile

Option	Description	Default
Step profile	Select a profile from the profile catalog. You can select a step profile when the **Step type** is set to custom and the **Part name/Joint**	`PL100*80` The default name is `STEP`.

Option	Description	Default
	no. option has not been set.	
Part name/ Joint no.	Select a custom part or a connection from the **Applications & components** catalog.	
Attribute file	Select the attribute file.	`standard`
Vertical position	Select the vertical depth position of a step that is created as a custom part. If you are using a custom connection, this option is not used.	`Middle`
Horizontal position	Select the horizontal on plane position of a step that is created as custom part. If you are using a custom connection, this option is not used.	`Middle`

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Bracket tab

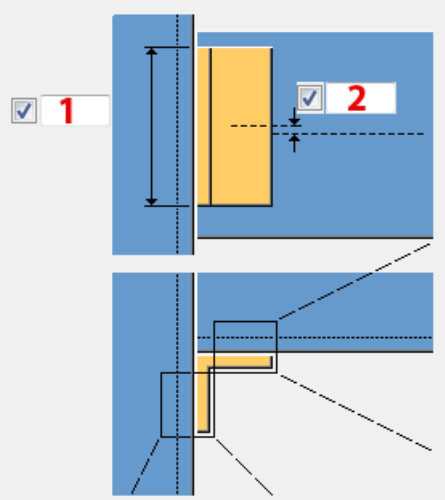
Use the **Bracket** tab to set up brackets, and the connection between the brackets and the steps, and the brackets and the stringers. The brackets support the angle profiles under the steps.

Bracket

Part	Description	Default
L profile	To create an L profile, select a profile from the profile catalog.	`BLL80*80*10` The default name is `L_Profile`.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

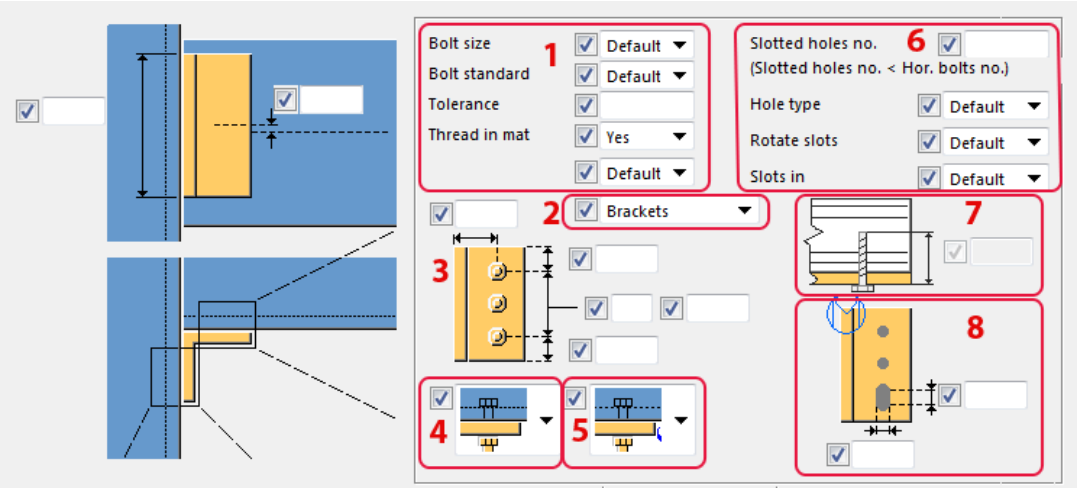
Bracket dimensions



	Description
1	Define the length of the bracket. The default length is calculated according to the bracket-to-step or bracket-to-stringer bolt dimensions, depending on which are bigger.
2	Define the bracket offset dimension from the center line of the step. The default is `0 mm`.

Bracket-to-step connection

Define the properties of the bolts that connect the bracket to the step.



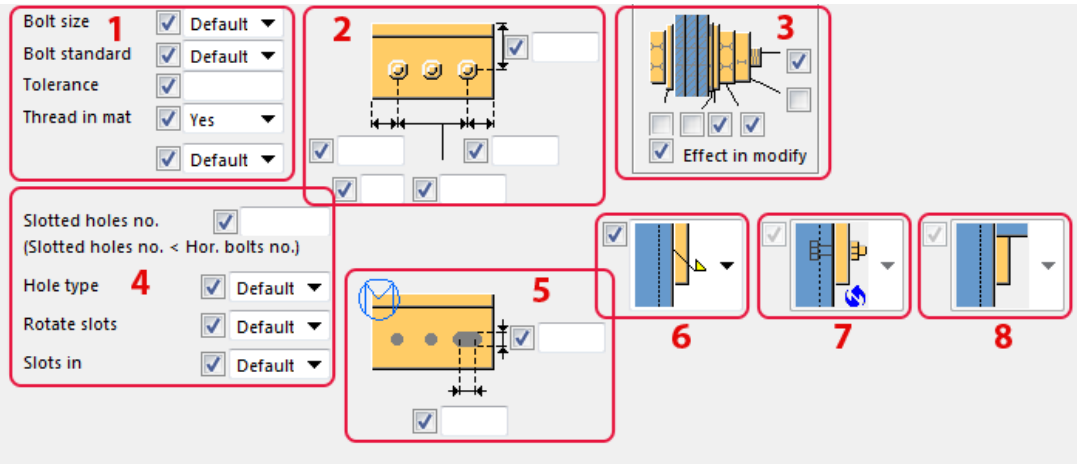
	Description
1	Define the basic bolt basic properties: • **Bolt size** defines the bolt diameter. Available sizes are defined in the bolt assembly catalog.

	Description
	• **Bolt standard** defines the bolt standard to be used inside the component. Available standards are defined in the bolt assembly catalog. • **Tolerance** defines the gap between the bolt and the hole. • **Thread in mat** defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.
2	Select whether brackets are created or not.
3	Define the bolt group dimensions.
4	Select the connection type between the steps and the brackets: • Bolted Bolted is the default. • Welded • No connection • Plate bracket Bracket angle leg is not created. • Holes only
5	Select the bolt direction.
6	Define the slotted hole properties: • **Slotted holes no.** defines the number of slotted holes. • **Hole type** creates either **Slotted** holes, or **Oversized** or tapped holes. • **Rotate slots** rotates slots when the hole type is slotted. • **Slots in** defines in which parts the slotted holes are created.

	Description
7	Define the exact length of the bolts. You can define the length of the bolts if the **Step type** is set to on the **Steps** tab.
8	Define the slotted hole dimensions.

Bracket-to-stringer connection

Define the properties of the bolts that connect the bracket to the stringer.



	Description
1	Define the basic bolt properties: • **Bolt size** defines the bolt diameter. Available sizes are defined in the bolt assembly catalog. • **Bolt standard** defines the bolt standard to be used inside the component. Available standards are defined in the bolt assembly catalog. • **Tolerance** defines the gap between the bolt and the hole. • **Thread in mat** defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.
2	Define the bolt group dimensions.
3	Define the bolt assembl. The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly. If you want to create a hole only, clear all the check boxes. To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.
4	Define the slotted hole properties. • **Slotted holes no.** defines the number of slotted holes.

	Description
	• **Hole type** creates either **Slotted** holes, or **Oversized** or tapped holes. • **Rotate slots** rotates slots when the hole type is slotted. • **Slots in** defines in which parts the slotted holes are created.
5	Define the slotted hole dimensions.
6	Select the connection type between the stringer and the brackets: • Bolted Bolted is the default. • Welded • No connection
7	Select the bolt direction.
8	Set the bracket position of a plate bracket: • Below the step Below the step is the default. • Between the step and the stringer You can use this option when you have set the connection type between the steps and the brackets to the plate bracket option .

Welds

Click the link below to find out more:

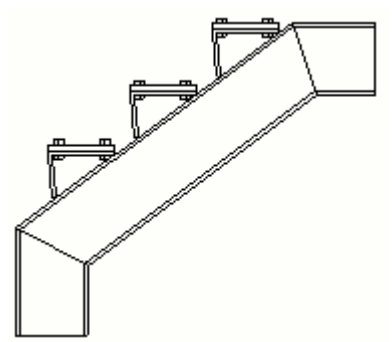
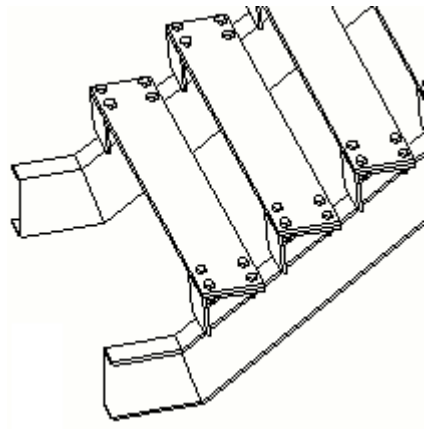
Wooden steps pan (S72)

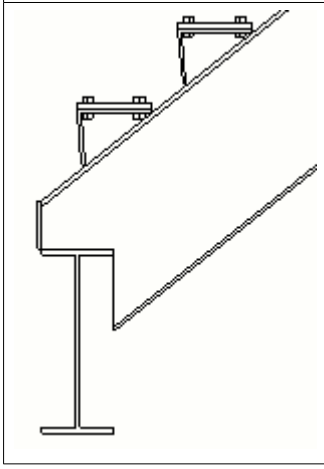
Wooden steps pan (S72) creates straight stairs with optional top and bottom landings. The stairs consist of stringers, possible landings and the actual steps.

Objects created

- Stringers
- Steps
- Landings (optional)
- Brackets
- Plates to the stringer ends (optional)
- Bolts
- Welds

Use for

Situation	Description
	Stairs with wooden steps. Vertical bottom landing type.
	Stairs with C-shaped stringers. Horizontal bottom landing type. Wooden steps are bolted to steel brackets.

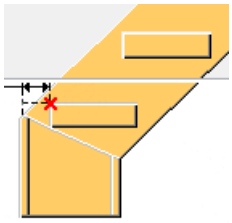
Situation	Description
	Stairs with stringers notched to the supporting beam.

Before you start

If the stringers are notched to bear on the supporting beams, create the supporting beams before creating the stairs.

Selection order

1. Pick a point to indicate the nosing point of the first step.

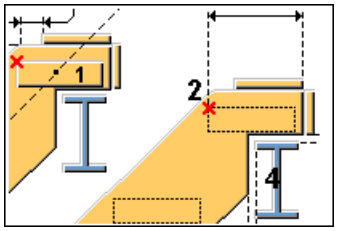


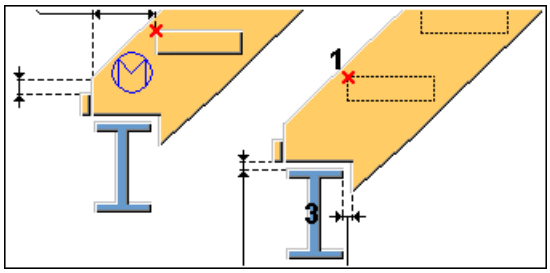
2. Pick another point to indicate the nosing point of the last step.

   The order in which the points are picked has no effect.

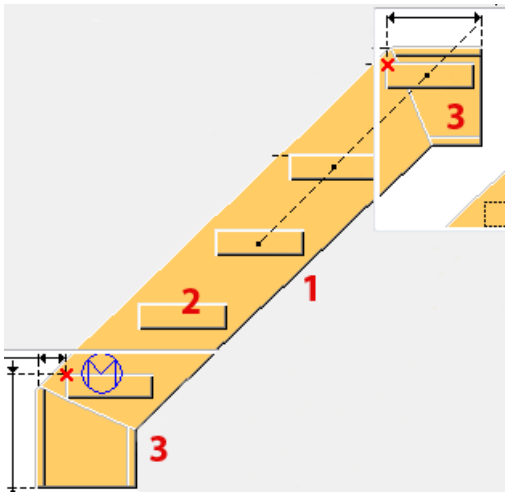
3. Click the middle mouse button to create the stairs.

   If you have selected the following notched options on the **Picture** tab, select the supporting beams and then click the middle mouse button.





Part identification key

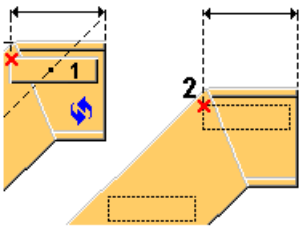


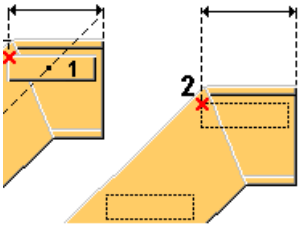
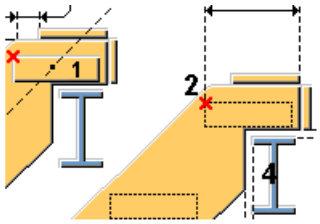
	Part
1	Stringer
2	Step
3	Landings

Picture tab

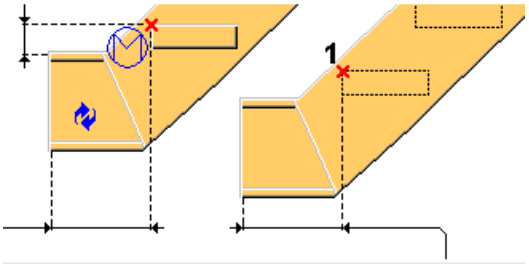
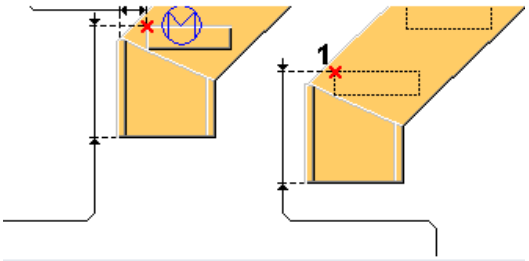
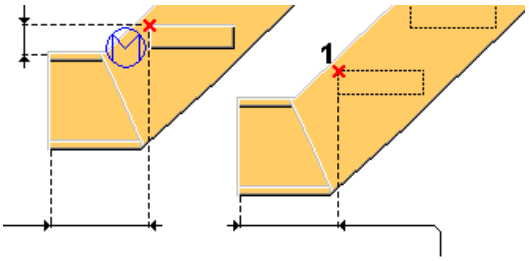
Use the **Picture** tab to control the stringer geometry and the nosing points of the steps.

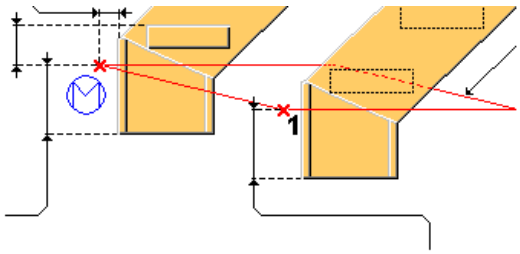
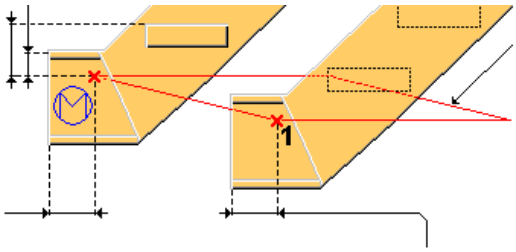
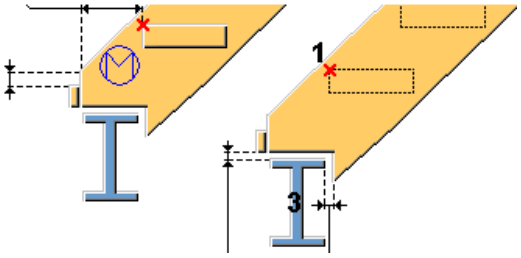
Top landing type

Option	Description
	Default

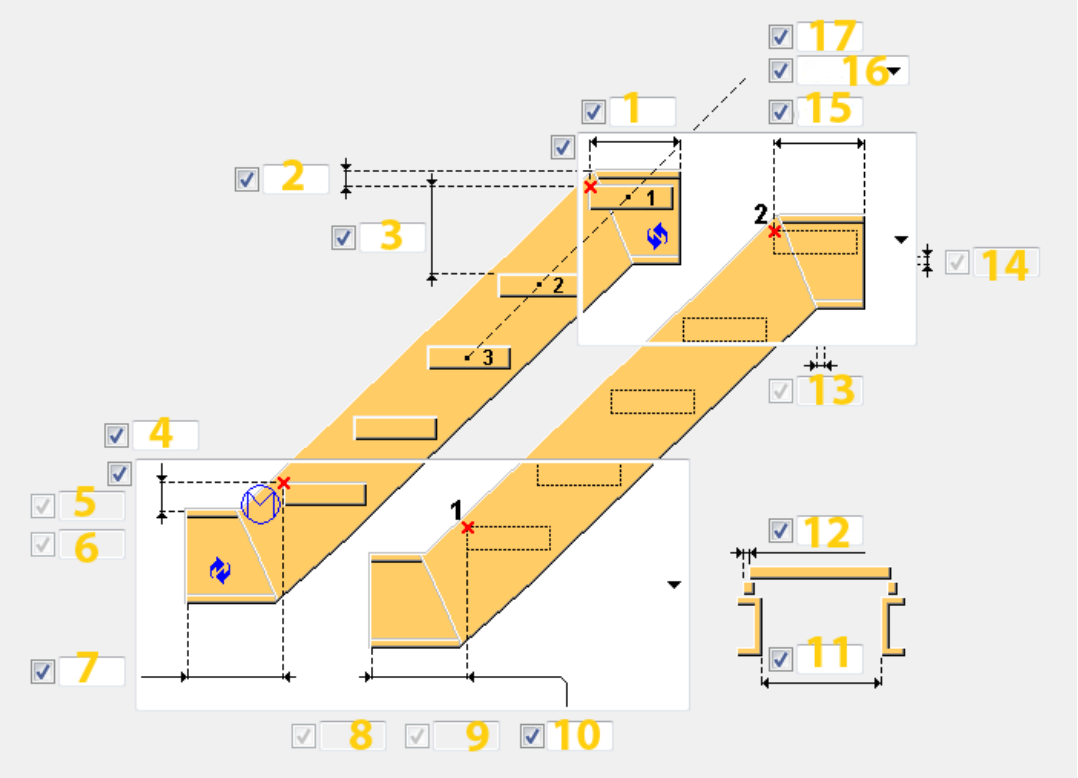
Option	Description
	Horizontal landing
	Notched The stringer top is on the supporting part. Vertical and horizontal plates are welded at the end of the stringer.

Bottom landing type

Option	Description
	Default
	Vertical Creates a vertical landing where the lower picked point is on the top plane of the last step.
	Horizontal Creates a horizontal landing where the lower picked point is on the top plane of the last step.

Option	Description
	Vertical lowered point The picked point is at the top of steel or finish floor.
	Horizontal lowered point The picked point is at the top of steel or finish floor.
	Notched Creates a notch to the bottom of the stringer to bear on the supporting part. A vertical plate is welded at the end of the stringer.

Stair dimensions



	Description	Default
1	Define the distance between the upper point and the end of the left top landing. If you set the value to 0 mm, the left top landing is not created. If you set the top landing type to notched, define the cut dimension of the horizontal plate from the stringer edge.	0 mm
2	Define the vertical distance from the upper point to the top of steel of the top landing. If the upper landing is not created, the stringer top of steel is used. If you set the top landing type to notched, define the vertical distance from the upper point to the stringer top horizontal cut.	200 mm

	Description	Default
3	Define the maximum distance allowed between two consecutive steps. The spacing between the steps is calculated using the step spacing type **Exact** or **Equal**, and the number of steps. For example, if the step height should be `200 mm`, the **Exact** setting creates steps of exactly this height (enter the number of steps as an integer). The **Equal** setting creates the required number of steps from the bottom to the top, with the height calculated as close to `200 mm` as possible.	`(Z*220)/(Z+220) mm`, where `Z` is the vertical distance between the 2 picked points.
4	Define this dimension based on the selected bottom landing type: • Horizontal or horizontal lowered point Define the vertical distance between the lower picked point and the top of steel of the bottom landing. • Vertical or vertical lowered point Define the horizontal distance between the lower picked point and the most distant face of the bottom landing. • No bottom landing is selected Define the horizontal distance from the lower picked point to the stringer vertical cut. By default, the cut is made at the supporting part edge.	`150 mm`
5	Define the vertical dimension from the lower point to the first step. You can define this dimension when the step spacing type is set to **Equal** and the bottom landing type is either vertical or horizontal lowered point.	`equal with vertical spacing between steps`
6	Define the cut dimension of the vertical plate from the stringer edge. You can define this dimension when the bottom landing type is set to notched.	`0 mm`

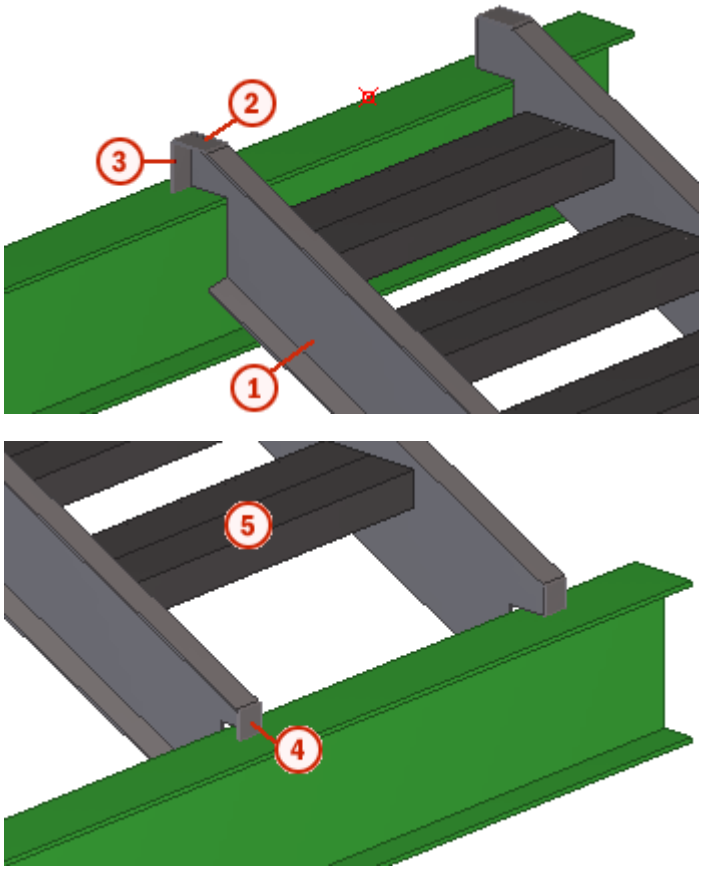
	Description	Default
7 **10**	Define this dimension based on the selected bottom landing type: • Horizontal or horizontal lowered point bottom landing. Define the horizontal edge distance between the lower point and the bottom landing. • Vertical or vertical lowered point bottom landing. Define the vertical edge distance between the lower point and the lowest point of the bottom landing.	600 mm
8	Define the vertical gap between the lower supporting part and the stringer cut. You can define this dimension when you have set the bottom landing type to notched.	0 mm
9	Define the horizontal gap between the lower supporting part and the stringer cut. You can define this dimension when you have set the bottom landing type to notched.	0 mm
11	Define the width of the steps.	0 mm
12	Define the dimension to shorten the steps equally on both sides.	1000 mm
13	Define the horizontal gap between the upper supporting part and the stringer cut. You can define this dimension when you have set the top landing type to notched.	0 mm
14	Define the vertical gap between the upper supporting part and the stringer cut. You can define this dimension when you have set the top landing type to notched.	0 mm
15	Define this dimension based on the selected top landing type: • Horizontal top landing. Define the distance between the upper point and the end of the stringer.	0 mm

	Description	Default
	• Notched top landing. Define the horizontal distance between the upper point and the stringer vertical cut. By default the cut is made at the supporting part edge. If you set this dimension to `0 mm`, the top landing is not created.	
16	Select the step spacing type: • **Exact** creates the steps of exactly the defined height. Enter the number of steps as an integer. • **Equal** creates the required number of steps from the bottom to the top, with the height calculated as close as possible to the defined step height. The spacing type controls how the vertical spacing between the steps is calculated. If you select **Exact** and set the bottom landing type to horizontal or vertical lower point, the steps are spaced from the upper point to the lower point. In all other cases the steps are spaced from the lower point to the upper point.	`Equal`
17	Enter the number of steps. This option depends on step spacing type and step spacing. If you set the step spacing type to **Exact**, or to **Equal** and the spacing distance is not defined, **Wooden steps pan (S72)** creates the number of steps you have defined.	`Calculated from the vertical distance between the 2 picked points and the spacing between steps (3)`

Stair setup tab

Use the **Stair setup** tab to control the part properties, top and bottom steps, position of the stairs on the horizontal plane, and rotation of stringers and steps.

Stair part properties



	Part	Description	Default
1	**Left stringer**	Always created Select a profile from the profile catalog.	BLU400*2 The default name is STRINGER.
1	**Right stringer**	Always created Select a profile from the profile catalog.	BLU400*2 The default name is STRINGER.
2	**Upper H plate**	Created only if the top of the stringer is on the supporting beam.	6 mm The default name is PLATE.
3	**Upper V plate**	Created only if the top of the stringer in on the supporting beam.	6 mm The default name is PLATE.
4	**Lower V plate**	Created only if the bottom of the stringer in on the supporting beam.	6 mm The default name is PLATE.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

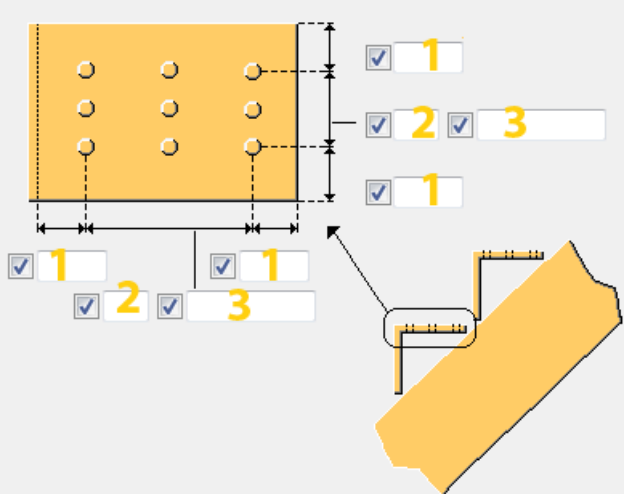
Stair setup settings

Option	Description
Create assembly	Define which parts of the stair component form an assembly. The default is stringers.
Stringer reference line	• **Above**: The line positioned on the inside of the stairs on the top face is used as the reference line. • **Nosing**: The nosing line of the steps is used as the reference line. The reference line is used for workshop drawings.
Steps rotation	Rotation of the step around its axis. The default is **Top**.
Position in plane	Position of the stairs. The middle line of steps is used as the reference line. The default is **Right**.
Offset	Offset of the stairs on the plane from the position that is set in the **Position in plane** option. The default offset is `0 mm`.
Stringer rotation	Rotation of the stringer around its axis. The default is top .
Create top step	Define whether the first step of the stairs (the highest step) is created. By default, the first step is created.
Create bottom step	Define whether the last step of the stairs (the lowest step) is created. By default, the last step is created.

Bolts tab

Use the **Bolts** tab to control the properties of the bolt group that connects the wooden steps to the brackets.

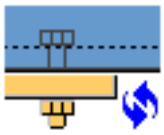
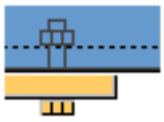
Bolt group dimensions

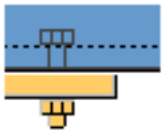


	Description	Default
1	Bolt edge distance.	`bolt diameter * 1.5`
2	Number of bolts.	`2`
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.	`100 mm`

Bolting direction

Use to change the direction of bolts.

Option	Description
	Default.
	Bolted from the bracket to the step.

Option	Description
	Bolted from the step to the bracket.

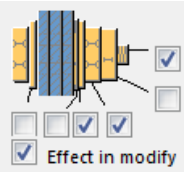
Bolt basic properties

	Description	Default
Bolt size	Bolt diameter.	`20 mm`
Bolt standard	The bolt standard to be used inside the component.	`7990`
Tolerance	The gap between the bolt and the hole.	`2 mm`
Thread in mat	Defines whether or not the thread may be within the bolted parts when using bolts with a shaft. This has no effect when using full-threaded bolts.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Wooden pan tab

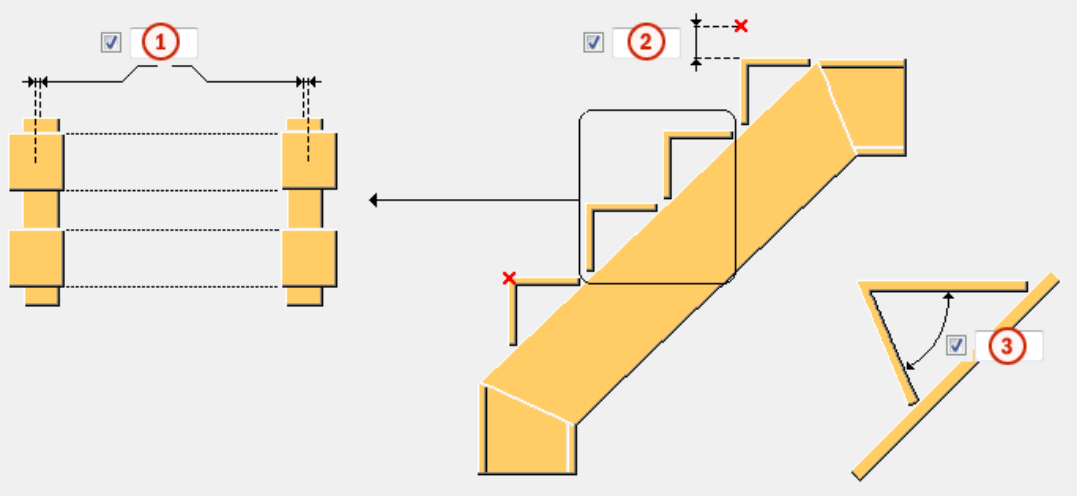
Use the **Wooden pan** tab to define the step profile and the properties of the folded plate brackets.

Step bracket and profile

Part	Description	Default
Step bracket	To create a step bracket, select a profile from the profile catalog.	PL10*150
Step profile	To create steps, select a profile from the profile catalog.	

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Bracket dimensions



	Description	Default
1	Define the offset dimension that moves the folded bracket plates relative to the center line of the stringer.	0 mm
2	Define the thickness of the steps that will be placed on the folded brackets. The actual steps will not be created, only the brackets height will be resized.	0 mm
3	Define the angle for the bracket to be folded, in degrees.	0

Welds

Click the link below to find out more:

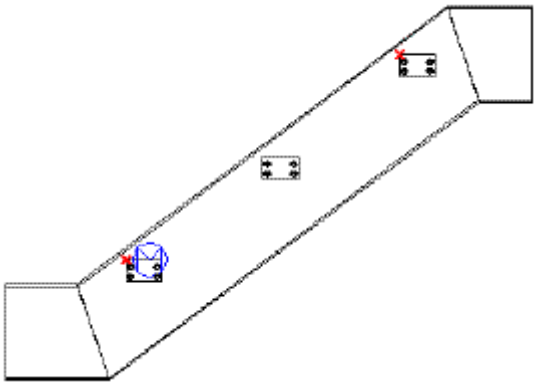
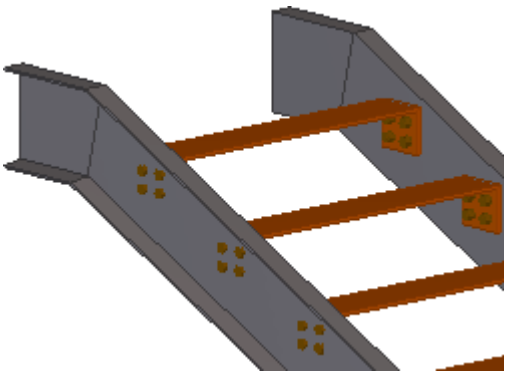
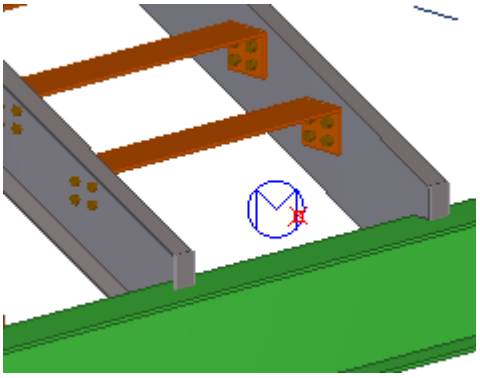
Polybeam pan (S73)

Polybeam pan (S73) creates straight stairs with optional top and bottom landings. The stairs consist of stringers, possible landings, and the actual steps created with plates.

Objects created

- Stringers
- Steps
- Landings (optional)
- Cuts to notch the stringer (optional)
- Plates (optional)
- Bolts
- Welds

Use for

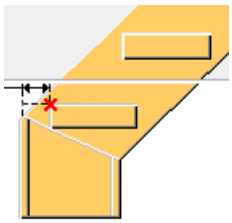
Situation	Description
	Polybeam pan stairs.
	Top of the stringer is created as horizontal landing.
	Bottom of the stringer is notched to bear on the supporting beam. The same option is also available for the top of the stringer.

Before you start

If the stringers are notched to bear on the supporting beams, create the supporting beams before creating the stairs.

Selection order

1. Pick a point to indicate the nosing point of the first step.

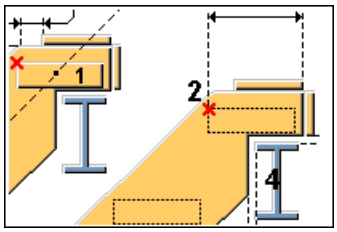


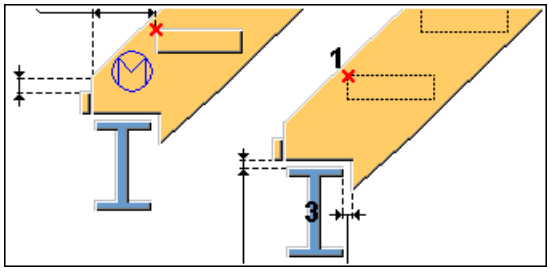
2. Pick another point to indicate the nosing point of the last step.

   Note that the order in which the points are picked has no effect.

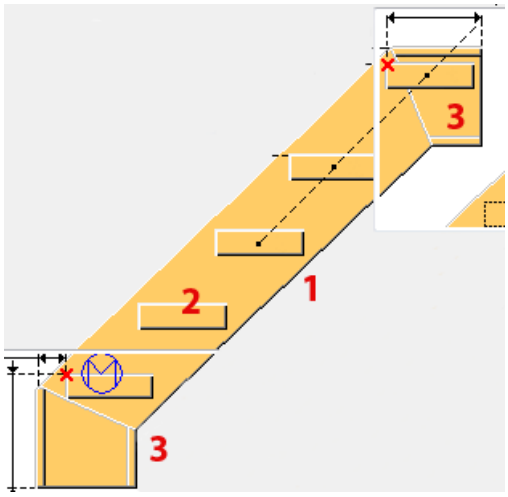
3. Click the middle mouse button to create the stairs.

   If you have selected the following notched options on the **Picture** tab, select the supporting beams and then click the middle mouse button.





Part identification key

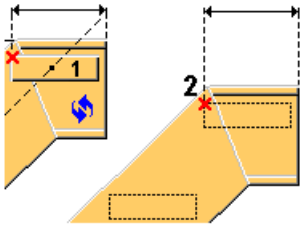
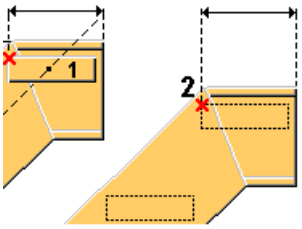
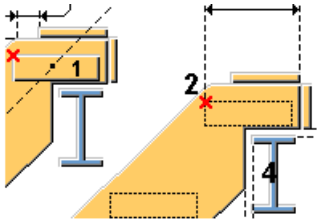


	Part
1	Stringer
2	Step
3	Landings

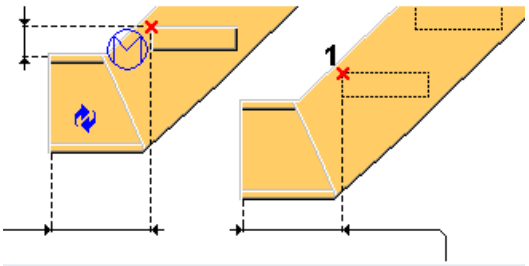
Picture tab

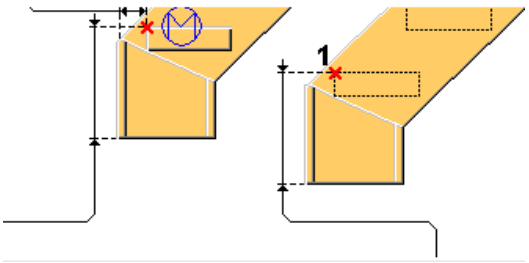
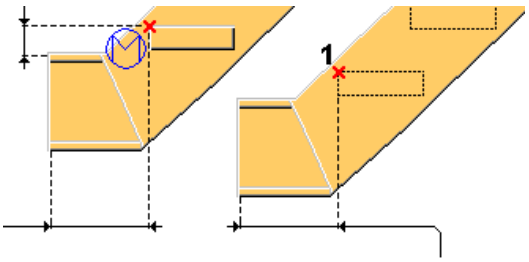
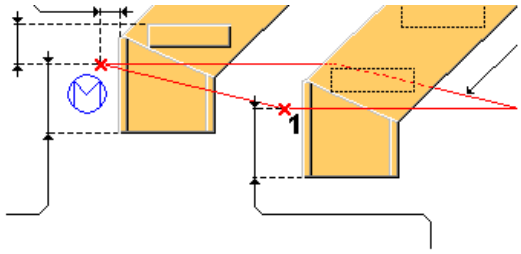
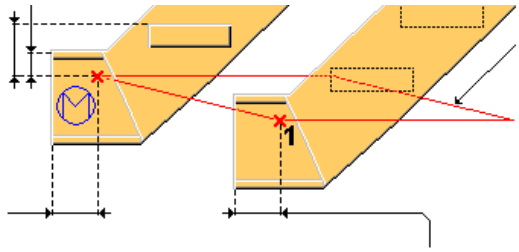
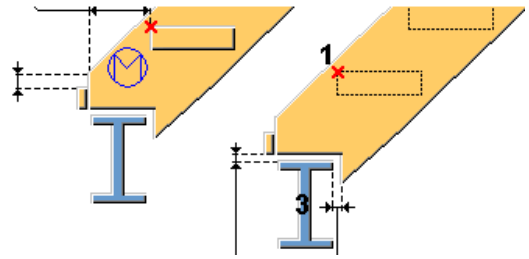
Use the **Picture** tab to control the stringer geometry and the nosing points of the steps.

Top landing type

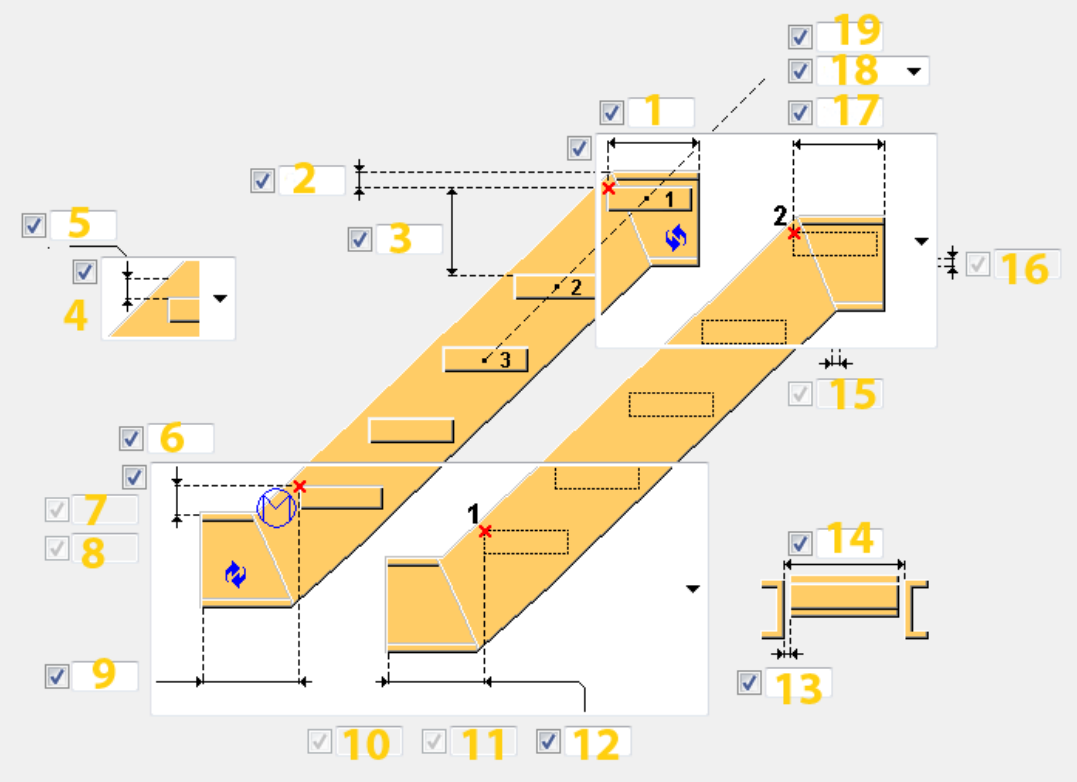
Option	Description
	Default
	Horizontal landing
	Notched The stringer top is on the supporting part. Vertical and horizontal plates are welded at the end of the stringer.

Bottom landing type

Option	Description
	Default

Option	Description
	Vertical Creates a vertical landing where the lower picked point is on the top plane of the last step.
	Horizontal Creates a horizontal landing where the lower picked point is on the top plane of the last step.
	Vertical lowered point The picked point is at the top of steel or finish floor.
	Horizontal lowered point The picked point is at top of steel or finish floor.
	Notched Created a notch to the bottom of the stringer to bear on the supporting part. A vertical plate is welded at the end of the stringer.

Stair dimensions



	Description	Default
1	Define the distance between the upper point and the end of the left top landing. If you set the value to `0 mm`, the left top landing is not created. If you set the top landing type to notched, define the cut dimension of the horizontal plate from the stringer edge.	`0 mm`
2	Define the vertical distance from the upper point to the top of steel of the top landing. If the top landing is not created, the stringer top of steel is used. If you set the top landing type to notched, define the vertical distance from the upper point to the stringer top horizontal cut.	`200 mm`

	Description	Default
3	Define the maximum distance allowed between two consecutive steps. The spacing between the steps is calculated using the step spacing type **Exact** or **Equal**, and the number of steps. For example, if the step height should be `200 mm`, the **Exact** setting creates steps of exactly this height (enter the number of steps as an integer). The **Equal** setting creates the required number of steps from the bottom to the top, with the height calculated as close to `200 mm` as possible.	`(Z*220)/(Z+220) mm`, where Z is the vertical distance between the 2 picked points.
4	Select the direction of the nosing dimension of the steps: • Vertical • Horizontal • Perpendicular	Vertical
5	Define the nosing dimension of the steps. This dimension depends on the direction you have selected.	`0 mm`
6	Define this dimension based on the selected bottom landing type: • Horizontal or horizontal lowered point bottom landing Define the vertical distance from the lower point to the top of steel of the bottom landing. • Vertical or vertical lowered point bottom landing Define the horizontal distance from the lower picked point to the most distant face of the bottom landing. • Notched bottom landing Define the horizontal distance from the lower point to the stringer vertical cut. By default, the cut is made at the supporting part edge.	`150 mm`
7	Define the vertical dimension from the lower point to the first step. You can define this dimension when the step spacing type is set to **Equal** and the	Equal with vertical spacing between steps

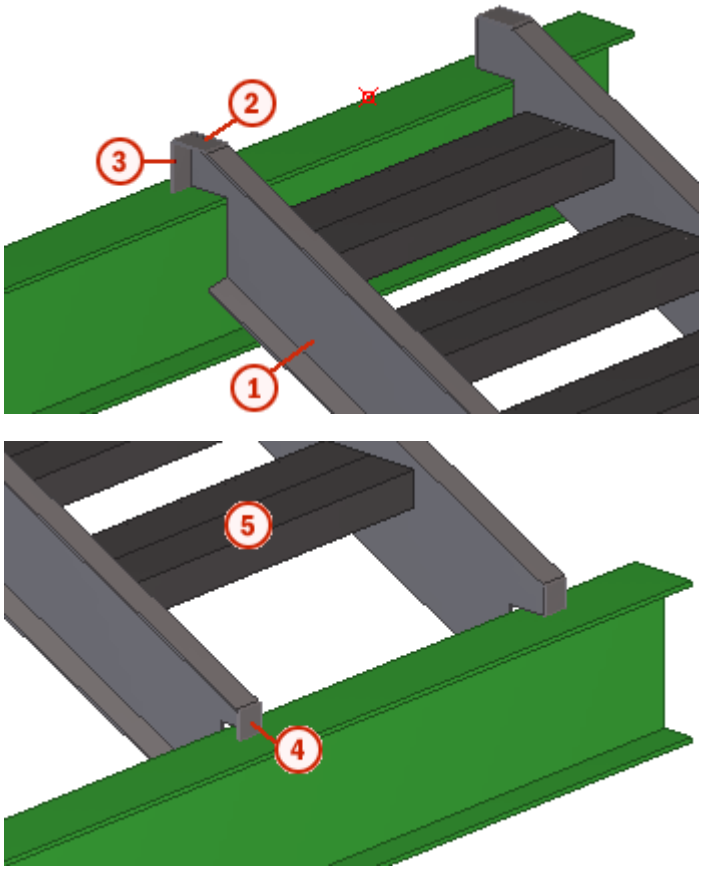
	Description	Default
	bottom landing type is either vertical or horizontal lowered point.	
8	Define the cut dimension of the vertical plate from the stringer edge. You can define this dimension when the bottom landing type is set to notched.	0 mm
9 **12**	Define this dimension based on the selected bottom landing type: • Horizontal or horizontal lowered point bottom landing. Define the horizontal edge distance between the lower point and the bottom landing. • Vertical or vertical lowered point bottom landing. Define the vertical edge distance between the lower point and the lowest point of the bottom landing.	600 mm
10	Define the vertical gap between the lower supporting part and the stringer cut. You can define this dimension when you have set the bottom landing type to notched.	0 mm
11	Define the horizontal gap between the lower supporting part and the stringer cut. You can define this dimension when you have set the bottom landing type to notched.	0 mm
13	Define the dimension to shorten the steps equally on both sides.	0 mm
14	Define the width of the steps.	1000 mm
15	Define the horizontal gap between the upper supporting part and the stringer cut. You can define this dimension when you have set the top landing type to notched.	0 mm
16	Define the vertical gap between the upper supporting part and the stringer cut. You can define this dimension when you have set the top landing type to notched.	0 mm

	Description	Default
17	Define this dimension based on the selected top landing type: • Horizontal top landing. Define the distance between the upper point and the end of the stringer. • Notched top landing. Define the horizontal distance between the upper point and the stringer vertical cut. By default the cut is made at the supporting part edge. If you set this dimension to `0 mm`, the top landing is not created.	`0 mm`
18	Select the step spacing type: • **Exact** creates the steps of exactly the defined height. Enter the number of steps as an integer. • **Equal** creates the required number of steps from the bottom to the top, with the height calculated as close as possible to the defined step height. The spacing type controls how the vertical spacing between the steps is calculated. If you select **Exact** and set the bottom landing type to horizontal or vertical lower point, the steps are spaced from the upper point to the lower point. In all other cases the steps are spaced from the lower point to the upper point.	**Equal**
19	Enter the number of steps. This option depends on step spacing type and step spacing. If you set the step spacing type to **Exact**, or to **Equal** and the spacing distance is not defined, **Polybeam pan (S73)** creates the number of steps you have defined.	Calculated from the vertical distance between the picked points and the spacing between the steps

Stair setup tab

Use the **Stair setup** tab to control the part properties, top and bottom steps, position of the stairs on the horizontal plane, and rotation of stringers and steps.

Stair part properties



	Part	Description	Default
1	**Left stringer**	Always created Select a profile from the profile catalog.	BLU400*2 The default name is STRINGER.
1	**Right stringer**	Always created Select a profile from the profile catalog.	BLU400*2 The default name is STRINGER.
2	**Upper H plate**	Created only if the top of the stringer is on the supporting beam.	6 mm The default name is PLATE.
3	**Upper V plate**	Created only if the top of the stringer in on the supporting beam.	6 mm The default name is PLATE.
4	**Lower V plate**	Created only if the bottom of the stringer in on the supporting beam.	6 mm The default name is PLATE.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

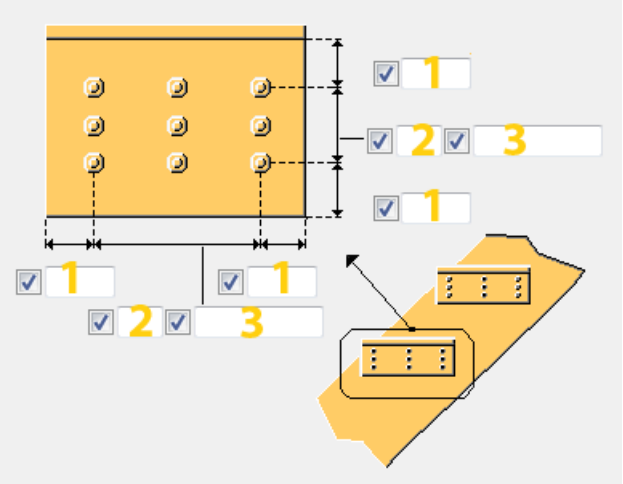
Stair setup settings

Option	Description
Create assembly	Define which parts of the stair component form an assembly. The default is stringers.
Stringer reference line	• **Above**: The line positioned on the inside of the stairs on the top face is used as the reference line. • **Nosing**: The nosing line of the steps is used as the reference line. The reference line is used for workshop drawings.
Steps rotation	Rotation of the step around its axis. The default is **Top**.
Position in plane	Position of the stairs. The middle line of steps is used as the reference line. The default is **Right**.
Offset	Offset of the stairs on the plane from the position that is set in the **Position in plane** option. The default offset is `0 mm`.
Stringer rotation	Rotation of the stringer around its axis. The default is top .
Create top step	Define whether the first step of the stairs (the highest step) is created. By default, the first step is created.
Create bottom step	Define whether the last step of the stairs (the lowest step) is created. By default, the last step is created.

Bolts tab

Use the **Bolts** tab to control the properties of the bolt group that connects the steps to the stringers.

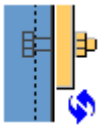
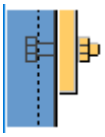
Bolt group dimensions

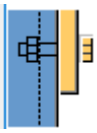


Note that if you define the vertical dimensions of the step on the **Polybeam pan** tab, the dimensions defined on the **Polybeam pan** tab are used.

	Description	Default
1	Bolt edge distance.	`bolt diameter * 1.5`
2	Number of bolts.	`2`
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.	`100 mm`

Bolting direction

Option	Description
	Default. Bolted from the step to the bracket. AutoDefaults can change this option.
	Bolted from the step to the bracket.

Option	Description
	Bolted from the bracket to the step.

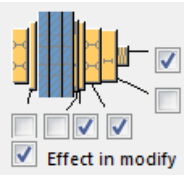
Bolt basic properties

	Description	Default
Bolt size	Bolt diameter.	`20 mm`
Bolt standard	The bolt standard to be used inside the component.	`7990`
Tolerance	The gap between the bolt and the hole.	`2 mm`
Thread in mat	Defines whether or not the thread may be within the bolted parts when using bolts with a shaft. This has no effect when using full-threaded bolts.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

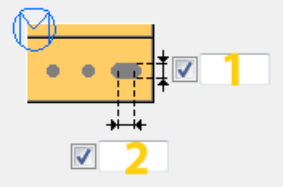
If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Slotted holes no	Define the number of slotted holes.	
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Polybeam pan tab

Use the **Polybeam pan** tab to define the step profile properties and dimensions.

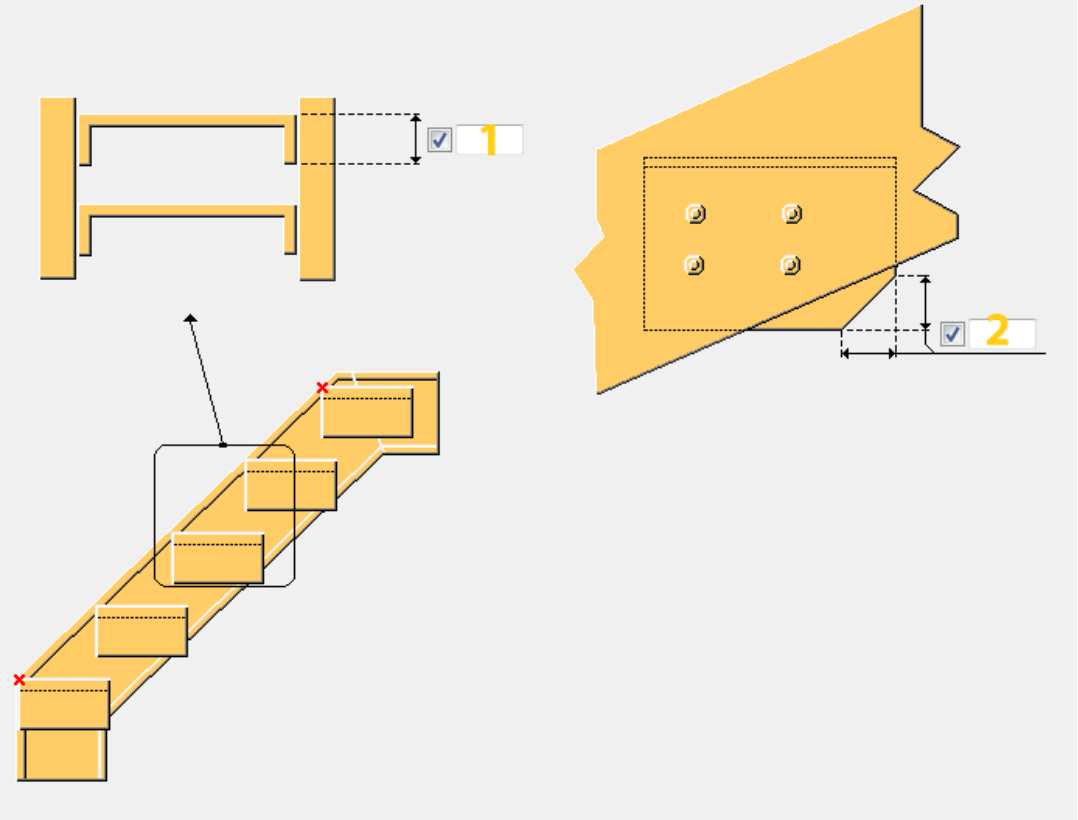
Plate

Part	Description	Default
Plate profile	To create a plate profile, select a profile from the profile catalog.	PL10*150

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
	assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Step profile dimensions



	Description	Default
1	Define the height of the vertical part of the step.	Vertical dimension value defined on the **Bolts** tab.
2	Define the symmetrical chamfer for the back corner of the all step plates.	0 mm

Welds

Click the link below to find out more:

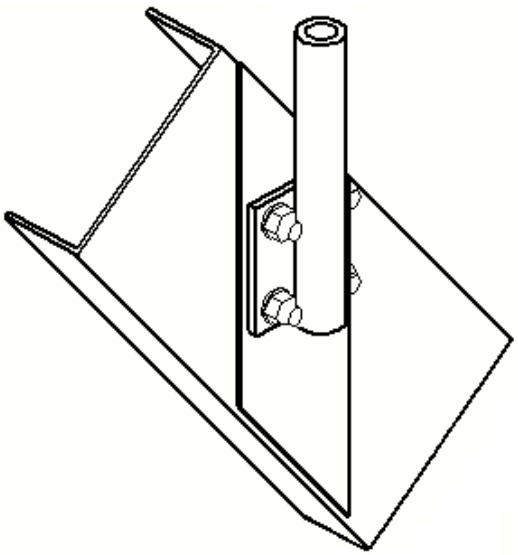
Handrail 1 (74)

Handrail 1 (74) connects a column to a beam using a cover plate and an end plate.

Objects created

- Cover plate
- End plate
- Bolts
- Welds
- Cuts

Use for

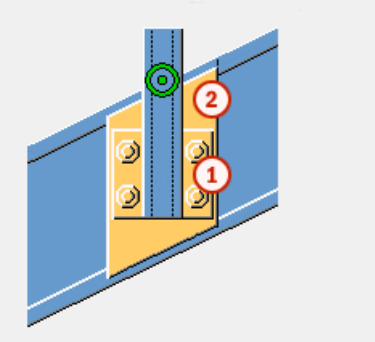
Situation	Description
	Beam to column connection.

Selection order

1. Select the main part (beam).
2. Select the secondary part (column).

   The connection is created automatically when the secondary part is selected.

Part identification key

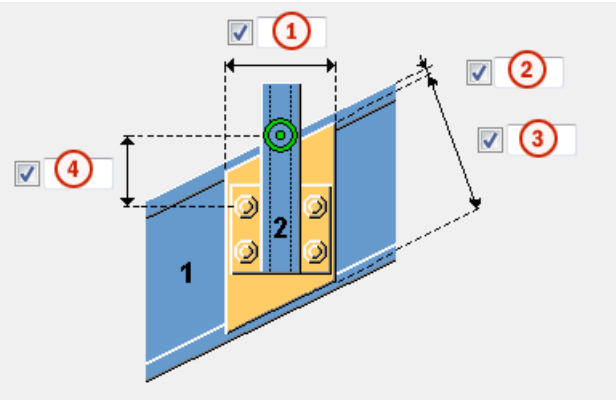


	Part
1	Cover plate
2	End plate

Picture tab

Use the **Picture** tab to control the plate dimensions.

Plate dimensions



	Description
1	Horizontal dimension of the cover plate.
2	Distance between the upper edge of the cover plate and the beam flange.
3	Vertical dimension of the cover plate.
4	Bolt edge distance.

Parts tab

Use the **Parts** tab to control the plate properties.

Plate

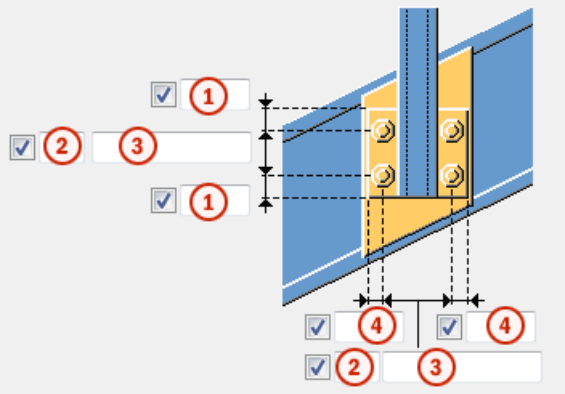
Option	Description
Cover plate	Cover plate thickness.
End plate	End plate thickness.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Bolts tab

Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions



	Description
1	Dimension for vertical bolt group position.
2	Number of bolts.

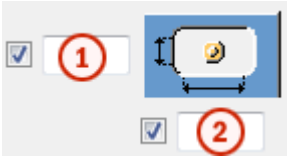
	Description
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Dimension for horizontal bolt group position.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



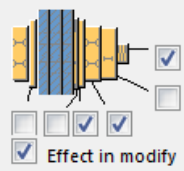
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

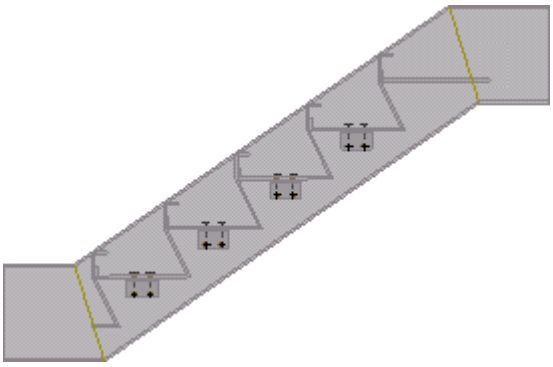
Z pan (S74)

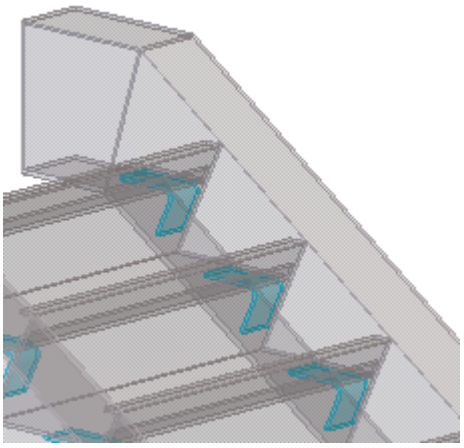
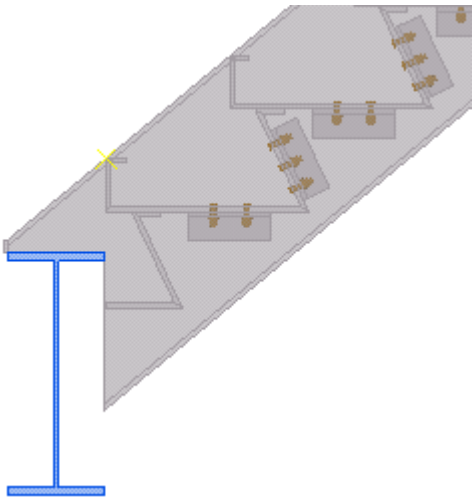
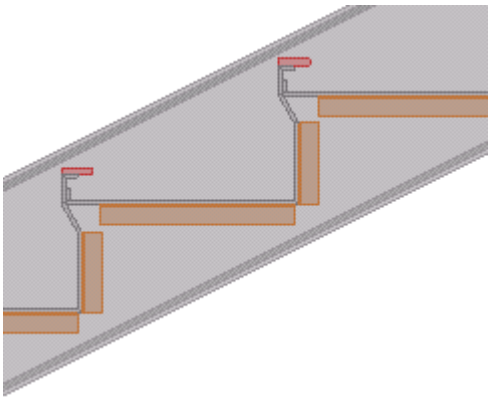
Z pan (S74) creates straight stairs with optional top and bottom landings. The stairs consist of stringers, possible landings, and the actual steps. The steps can be connected to stringers with horizontal brackets, vertical brackets, or with bent plate brackets.

Objects created

- Stringers
- Steps
- Landings (optional)
- Brackets
- Bolts
- Welds

Use for

Situation	Description
	Stairs with Z-pan steps. Horizontal top and bottom landings. Steps are connected to stringers with bolted horizontal brackets.

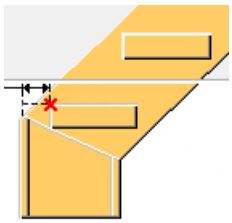
Situation	Description
	Stairs with bent plate brackets. Steps are connected to stringers with bent plate brackets.
	Stairs with stringers notched to the supporting beam. Steps are connected to stringers with vertical and horizontal brackets.
	Stairs with 90 degree bends at the top and bottom of the treads. Steps are connected to stringers with vertical and horizontal brackets. Red nose profile at the front edge of the steps.

Before you start

If the stringers are notched to bear on the supporting beams, create the supporting beams before creating the stairs.

Selection order

1. Pick a point to indicate the nosing point of the first step.

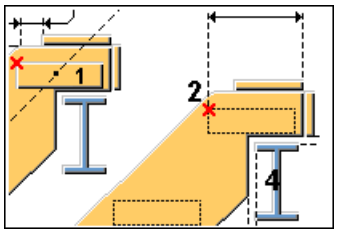


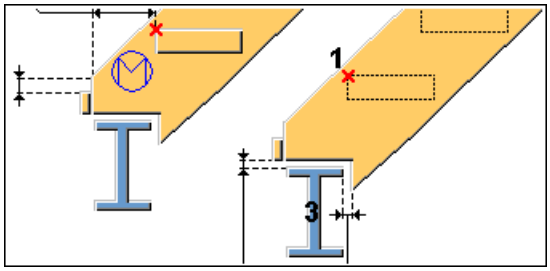
2. Pick another point to indicate the nosing point of the last step.

   Note that the order in which the points are picked has no effect.

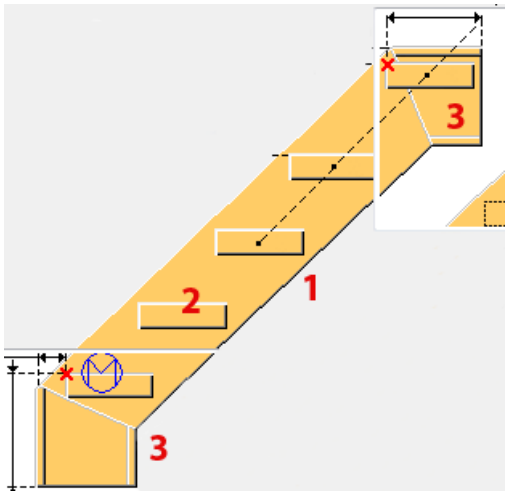
3. Click the middle mouse button to create the stairs.

   If you have selected the following notched options on the **Picture** tab, select the supporting beams and then click the middle mouse button.





Part identification key

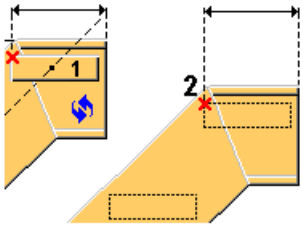
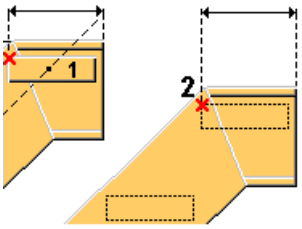
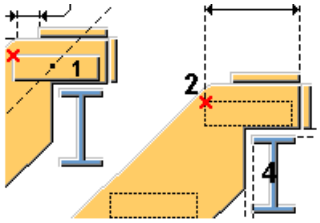


	Part
1	Stringer
2	Step
3	Landings

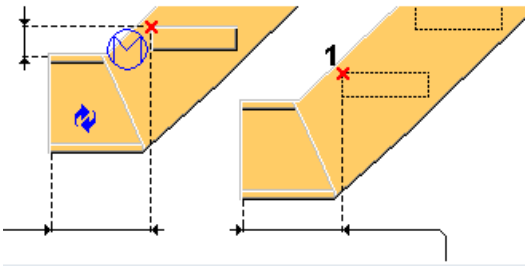
Picture tab

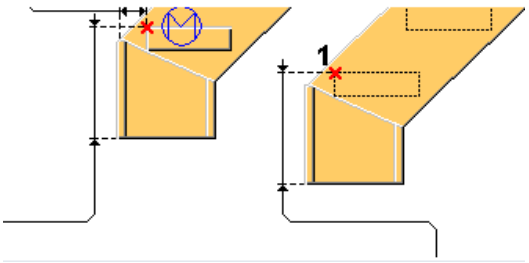
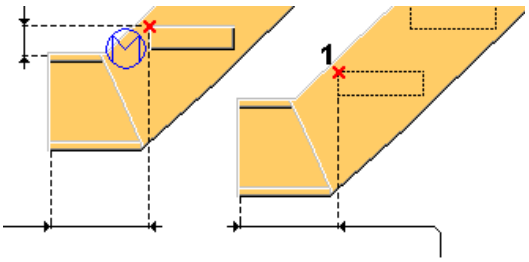
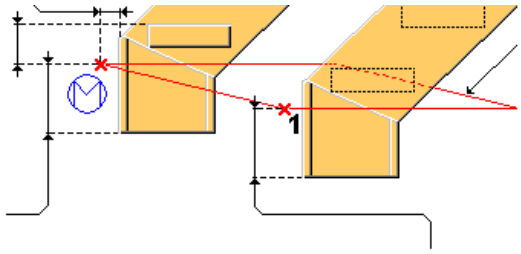
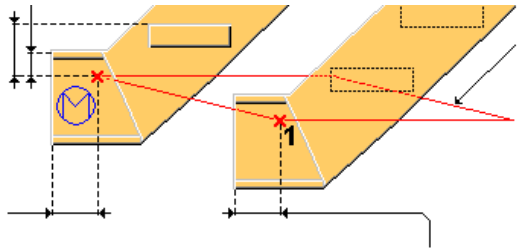
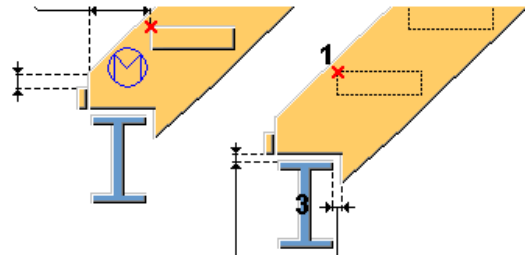
Use the **Picture** tab to control the stringer geometry and the nosing points of the steps.

Top landing type

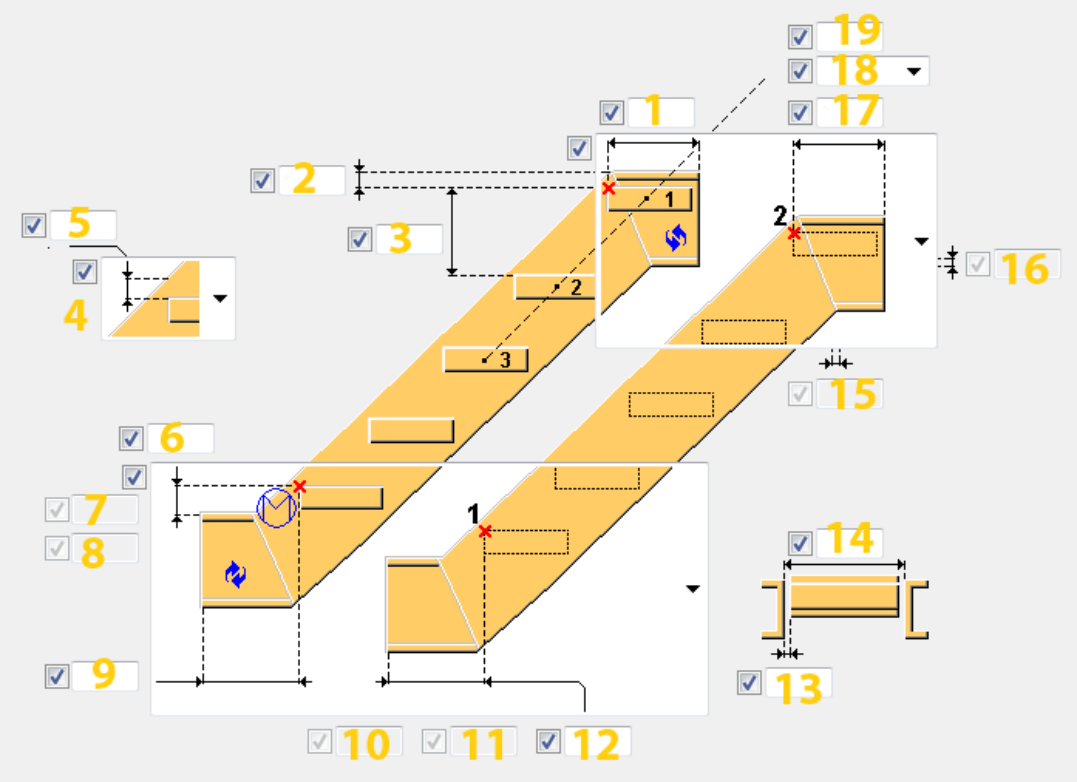
Option	Description
	Default
	Horizontal landing
	Notched The stringer top is on the supporting part. Vertical and horizontal plates are welded at the end of the stringer.

Bottom landing type

Option	Description
	Default

Option	Description
	Vertical Creates a vertical landing where the lower picked point is on the top plane of the last step.
	Horizontal Creates a horizontal landing where the lower picked point is on the top plane of the last step.
	Vertical lowered point The picked point is at the top of steel or finish floor.
	Horizontal lowered point The picked point is at top of steel or finish floor.
	Notched Creates a notch to the bottom of the stringer to bear on the supporting part. A vertical plate is welded at the end of the stringer.

Stair dimensions



	Description	Default
1	Define the distance between the upper point and the end of the left top landing. If you set the value to `0 mm`, the left top landing is not created. If you set the top landing type to notched, define the cut dimension of the horizontal plate from the stringer edge.	`0 mm`
2	Define the vertical distance from the upper point to the top of steel of the top landing. If the top landing is not created, the stringer top of steel is used. If you set the top landing type to notched, define the vertical distance from the upper point to the stringer top horizontal cut.	`200 mm`

	Description	Default
3	Define the maximum distance allowed between two consecutive steps. The spacing between the steps is calculated using the step spacing type **Exact** or **Equal**, and the number of steps. For example, if the step height should be `200 mm`, the **Exact** setting creates steps of exactly this height (enter the number of steps as an integer). The **Equal** setting creates the required number of steps from the bottom to the top, with the height calculated as close to `200 mm` as possible.	`(Z*220)/(Z+220) mm`, where Z is the vertical distance between the 2 picked points.
4	Select the direction of the nosing dimension of the steps: • Vertical • Horizontal • Perpendicular	Vertical
5	Define the nosing dimension of the steps. This dimension depends on the direction you have selected.	`0 mm`
6	Define this dimension based on the selected bottom landing type: • Horizontal or horizontal lowered point bottom landing Define the vertical distance from the lower point to the top of steel of the bottom landing. • Vertical or vertical lowered point bottom landing Define the horizontal distance from the lower picked point to the most distant face of the bottom landing. • Notched bottom landing Define the horizontal distance from the lower point to the stringer vertical cut. By default, the cut is made at the supporting part edge.	`150 mm`
7	Define the vertical dimension from the lower point to the first step. You can define this dimension when the step spacing type is set to **Equal** and the	Equal with vertical spacing between steps

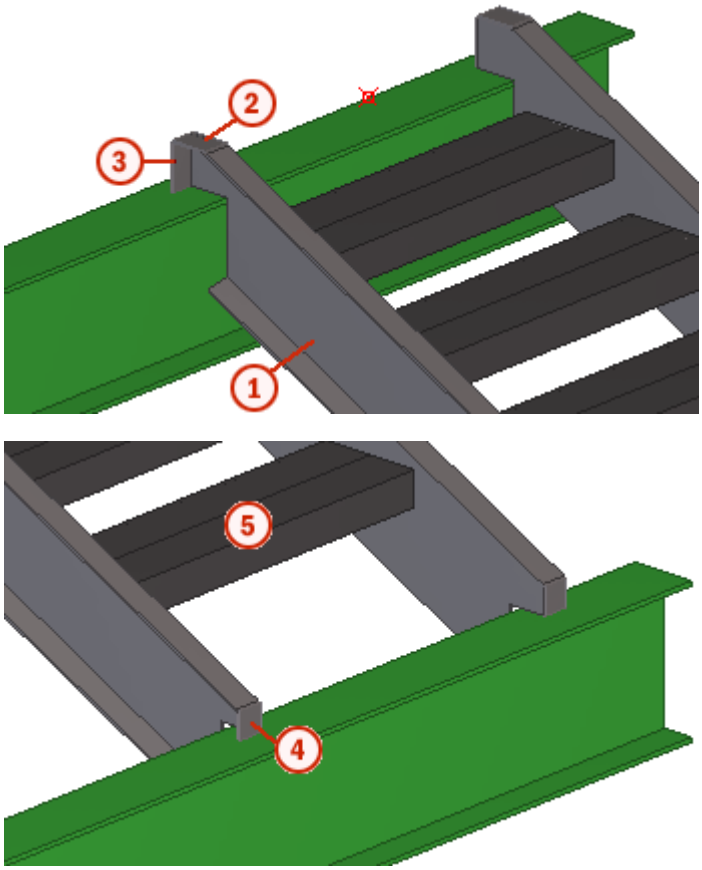
	Description	Default
	bottom landing type is either vertical or horizontal lowered point.	
8	Define the cut dimension of the vertical plate from the stringer edge. You can define this dimension when the bottom landing type is set to notched.	0 mm
9 **12**	Define this dimension based on the selected bottom landing type: • Horizontal or horizontal lowered point bottom landing. Define the horizontal edge distance between the lower point and the bottom landing. • Vertical or vertical lowered point bottom landing. Define the vertical edge distance between the lower point and the lowest point of the bottom landing.	600 mm
10	Define the vertical gap between the lower supporting part and the stringer cut. You can define this dimension when you have set the bottom landing type to notched.	0 mm
11	Define the horizontal gap between the lower supporting part and the stringer cut. You can define this dimension when you have set the bottom landing type to notched.	0 mm
13	Define the dimension to shorten the steps equally on both sides.	0 mm
14	Define the width of the steps.	1000 mm
15	Define the horizontal gap between the upper supporting part and the stringer cut. You can define this dimension when you have set the top landing type to notched.	0 mm
16	Define the vertical gap between the upper supporting part and the stringer cut. You can define this dimension when you have set the top landing type to notched.	0 mm

	Description	Default
17	Define this dimension based on the selected top landing type: • Horizontal top landing. Define the distance between the upper point and the end of the stringer. • Notched top landing. Define the horizontal distance between the upper point and the stringer vertical cut. By default the cut is made at the supporting part edge. If you set this dimension to `0 mm`, the top landing is not created.	`0 mm`
18	Select the step spacing type: • **Exact** creates the steps of exactly the defined height. Enter the number of steps as an integer. • **Equal** creates the required number of steps from the bottom to the top, with the height calculated as close as possible to the defined step height. The spacing type controls how the vertical spacing between the steps is calculated. If you select **Exact** and set the bottom landing type to horizontal or vertical lower point, the steps are spaced from the upper point to the lower point. In all other cases the steps are spaced from the lower point to the upper point.	**Equal**
19	Enter the number of steps. This option depends on step spacing type and step spacing. If you set the step spacing type to **Exact**, or to **Equal** and the spacing distance is not defined, **Z pan (S74)** creates the number of steps you have defined.	Calculated from the vertical distance between the picked points and the spacing between the steps

Stair setup tab

Use the **Stair setup** tab to control the part properties, top and bottom steps, position of the stairs on the horizontal plane, and rotation of stringers and steps.

Stair part properties



	Part	Description	Default
1	**Left stringer**	Always created Select a profile from the profile catalog.	BLU400*2 The default name is STRINGER.
1	**Right stringer**	Always created Select a profile from the profile catalog.	BLU400*2 The default name is STRINGER.
2	**Upper H plate**	Created only if the top of the stringer is on the supporting beam.	6 mm The default name is PLATE.
3	**Upper V plate**	Created only if the top of the stringer in on the supporting beam.	6 mm The default name is PLATE.
4	**Lower V plate**	Created only if the bottom of the stringer in on the supporting beam.	6 mm The default name is PLATE.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Stair setup settings

Option	Description
Create assembly	Define which parts of the stair component form an assembly. The default is stringers.
Stringer reference line	• **Above**: The line positioned on the inside of the stairs on the top face is used as the reference line. • **Nosing**: The nosing line of the steps is used as the reference line. The reference line is used for workshop drawings.
Position in plane	Position of the stairs. The middle line of steps is used as the reference line. The default is **Right**.
Offset	Offset of the stairs on the plane from the position that is set in the **Position in plane** option. The default offset is `0 mm`.
Stringer rotation	Rotation of the stringer around its axis. The default is top .
Create top step	Define whether the first step of the stairs (the highest step) is created. By default, the first step is created.
Create bottom step	Define whether the last step of the stairs (the lowest step) is created. By default, the last step is created.
Bracket	Select how the steps are connected to the stringers: • Default • Horizontal bracket

Option	Description
	• Horizontal and vertical brackets • Bent plate bracket

Z pan tab

Use the **Z pan** tab to control the size and shape of the Z pan steps.

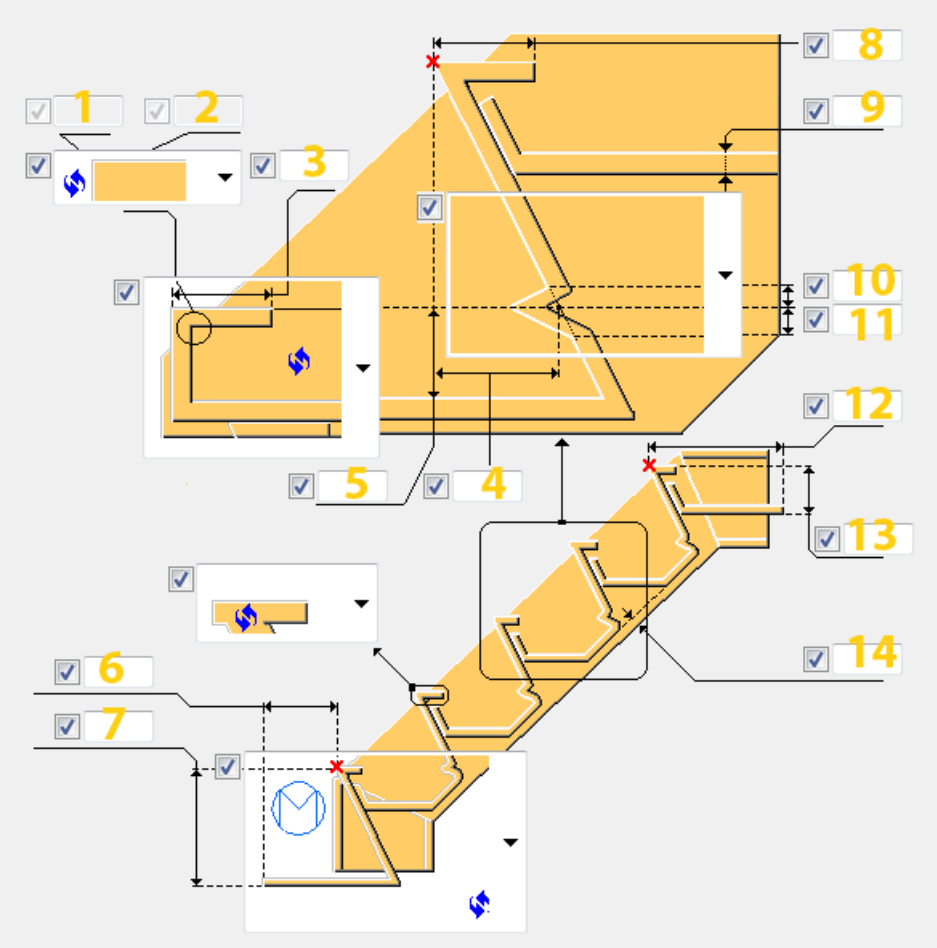
Profiles

Profile	Description	Default
Step profile	Step profile is calculated from the thickness you enter for the Z pan plate when defining the Z pan dimensions. Select an appropriate material for the step profile. In the US imperial environment, Tekla Structures saves the selected material in the **Gage material** user-defined attribute of the step profile. You can use this information in reports and in the bill of material of drawings.	
Nose profile	Created only if **Nosing piece** is set to **Create nose**. To create the nose profile, select the profile from the profile catalog.	`PL160*10`

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
	where you can enter the assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Z pan dimensions



	Description	Default
1 **2**	Define the chamfer dimensions for the front edge.	15 mm

	Description	Default
3	Define the length of the front part of the step according to the selected front edge type.	40 mm
4	Define back wall slope width between the nosing point vertical line and the Z pan back wall (concrete level lip set-off point).	100 mm
5	Define the height of the inner step dimension where the concrete will be poured.	100 mm
6 **7**	Define the width and height of the bottom step.	height = 200 mm width = 0 mm
8	Define the horizontal dimension for the Z pans that have a skewed front edge.	60 mm
9	Define the thickness of the Z pan plate.	10 mm
10 **11**	Define the lip dimensions of the back wall of the step. You can define these dimensions if **Back wall type** is set to **Lip**.	15 mm
12	Define the horizontal length of the top step. This is the horizontal distance between the top nosing point and the end of the top step horizontal section.	300 mm
13	Define the height of the top step. This is the vertical distance between the top nosing point and the end of the top step horizontal end.	100 mm
14	Define the bottom line clearance for the steps that prevent the bottom portion of the Z pan from overlapping the bottom end of the stringer.	20 mm

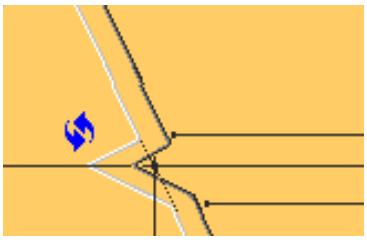
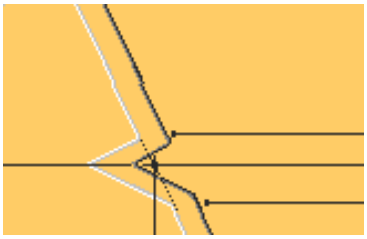
Chamfer type

Select the shape of the step front edge.

Option	Description
	None
	Line
	Rounding
	Arc

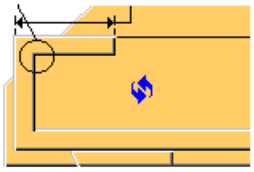
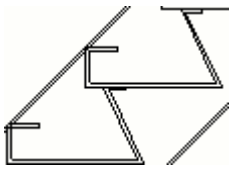
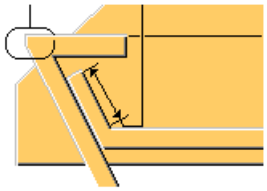
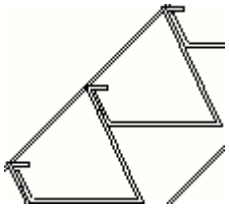
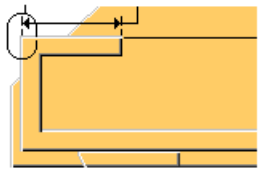
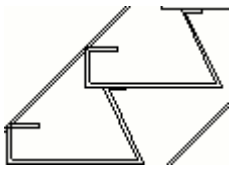
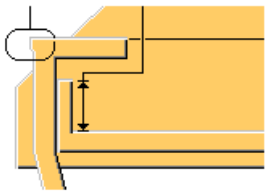
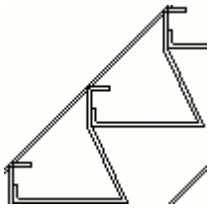
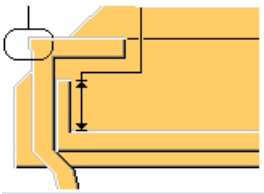
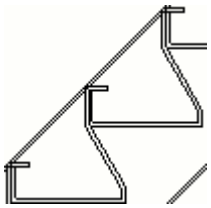
Back wall type

Select the Z pan back wall type.

Option	Description
	Default
	Lip
	Straight

Front edge type

Select how a pan connects to the pan below.

Option	Description	Example
	Default Rectangular front edge. AutoDefaults can change this option.	
	Skewed front edge. Pan overlaps the lower one and continues in the same angle as the back wall.	
	Rectangular front edge. Pans do not overlap each other.	
	Rectangular front edge. Pan overlaps the lower one and continues in the same angle as the back wall.	
	Rectangular front edge. Pan overlaps the lower one and continues at a 30-degree angle.	

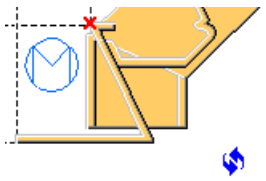
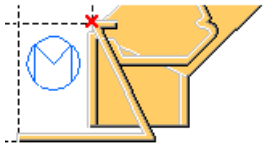
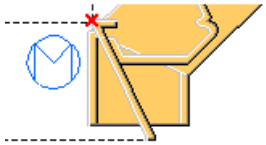
Nosing piece

Select whether the nosing piece is created on top of the step.

Option	Description
	Default No nose. AutoDefaults can change this option.
	No nose.
	Create nose.

Horizontal part

Select whether the bottom pan has a horizontal part. Enter the vertical distance from the bottom of the pan to the top level of the Z pan (concrete slab thickness) in Z pan dimensions.

Option	Description
	Default Create horizontal part. AutoDefaults can change this option.
	Create horizontal part.
	No horizontal part.

Horizontal bracket tab

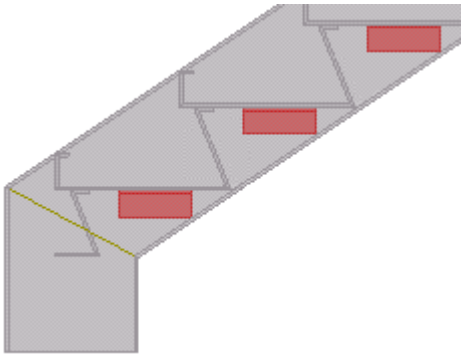
Use the **Horizontal bracket** tab to control the properties and attachment of the horizontal brackets. The horizontal brackets are angle profiles underneath the treads.

Preconditions

Before you can set the properties of the horizontal brackets, set the following option:

- On the **Stair set-up** tab, set **Bracket** to **Horizontal bracket** or **Horizontal and vertical brackets**.

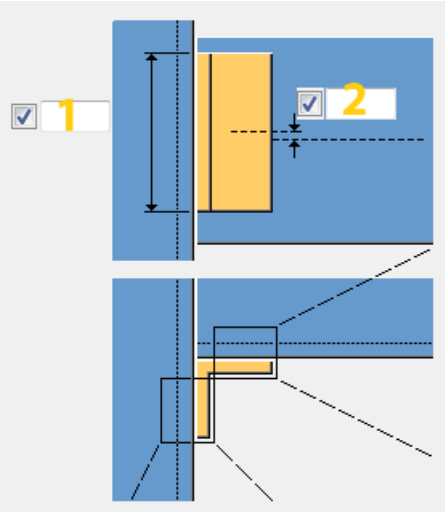
L profile



Part	Description	Default
L profile	Created only if brackets are created. Select the profile from the profile catalog.	BLL80*80*10

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Bracket dimensions

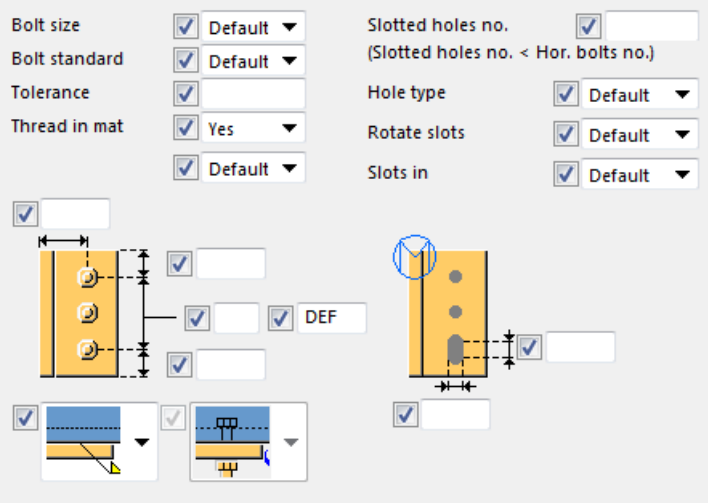


1	Define the length of the bracket. The default length is calculated according to bracket-to-step or bracket-to-stringer bolt dimensions, depending on which one is bigger.

2	Define the dimension to offset the bracket from the center line of the thread. The default in `0` mm.

Bracket-to-step connection

Define the properties of the bolts that connect the brackets to the steps.

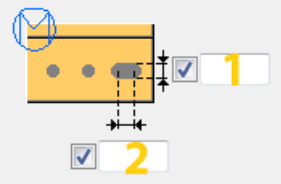


Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

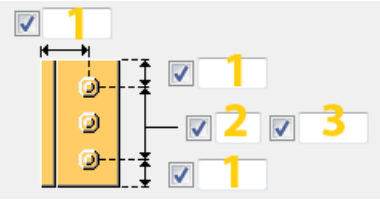
Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Slotted holes no	Define the number of slotted holes.	
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

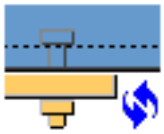
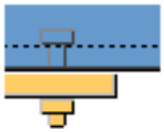
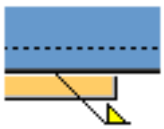
Bolt group dimensions



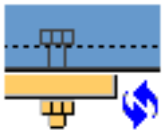
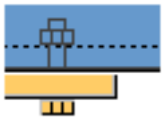
	Description	Default
1	Bolt edge distance.	`bolt diameter * 1.5`
2	Number of bolts.	`2`
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For	`100 mm`

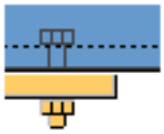
	Description	Default
	example, if there are 3 bolts, enter 2 values.	

Bracket-to-step connection type

Option	Description
	Default.
	Bolted.
	Welded.
	No connection.
	No bracket angle leg. The bracket is a plate instead of an angle.
	Holes only.

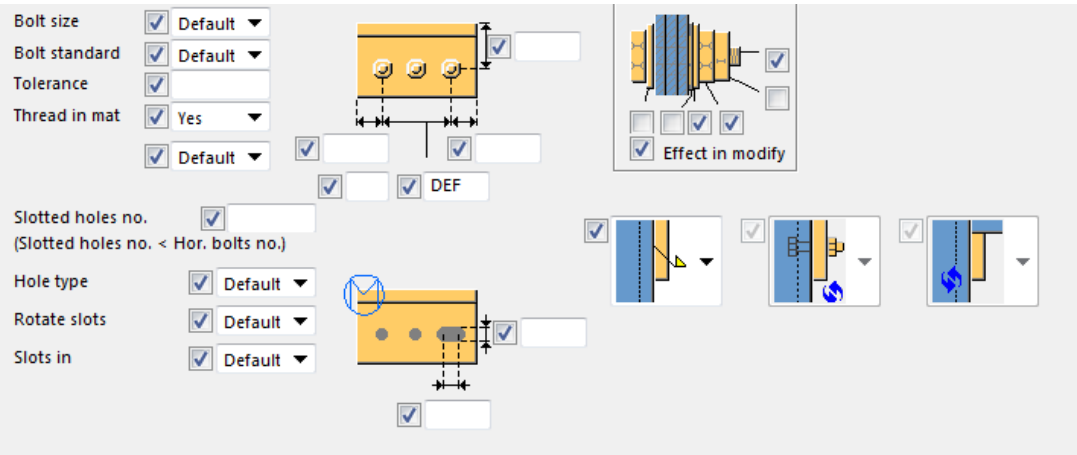
Bolting direction

Option	Description
	Default.
	Bolted from the bracket to the step.

Option	Description
	Bolted from the step to the bracket.

Bracket-to-stringer connection

Define the properties of the bolts that connect the brackets to the stringers.

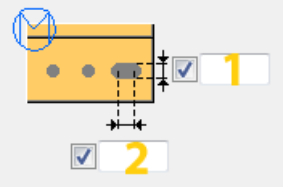


Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

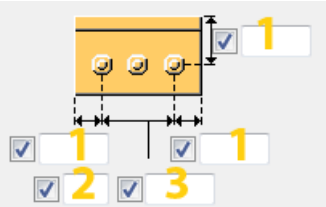
Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Slotted holes no	Define the number of slotted holes.	
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt group dimensions



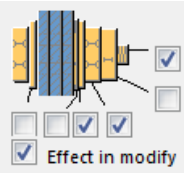
	Description	Default
1	Bolt edge distance.	`bolt diameter * 1.5`
2	Number of bolts.	`2`
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For	`100 mm`

	Description	Default
	example, if there are 3 bolts, enter 2 values.	

Bolt assembly

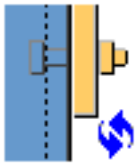
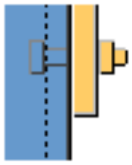
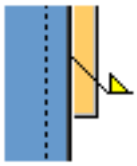
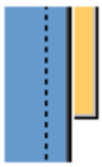
The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.

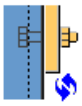
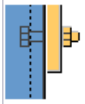
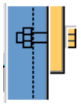


To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bracket-to-stringer connection type

Option	Description
	Default. Bolted. AutoDefaults can change this option.
	Bolted.
	Welded.
	No connection.

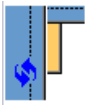
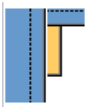
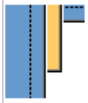
Bolting direction

Option	Description
	Default. Bolted from the bracket to the stringer. AutoDefaults can change this option.
	Bolted from the bracket to the stringer.
	Bolted from the stringer to the bracket.

Bracket position

Define the position of the plate-type bracket. The bracket can be positioned below the step, or between the step and the stringer.

You can define the bracket position if the bracket is a created as a plate instead of an angle.

Option	Description
	Default Below the step. AutoDefaults can change this option.
	Below the step.
	Between the stringer and step.

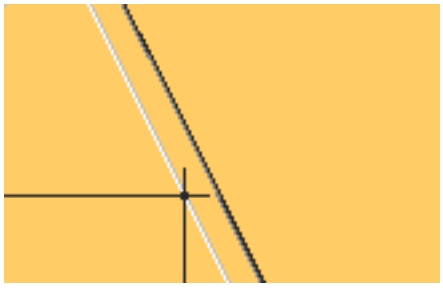
Vertical bracket tab

Use the **Vertical bracket** tab to control the properties and attachment of the vertical brackets. The vertical brackets are angle profiles underneath the treads and they support the vertical portion of the stair treads.

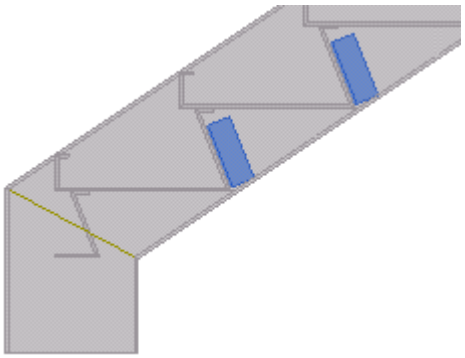
Preconditions

Before you can set the properties of the vertical brackets, set the following options:

- On the **Stair set up** tab, set **Bracket** to **Horizontal and vertical brackets**.
- On the **Z pan** tab, set the **Back wall type** to **Straight**.



L profile

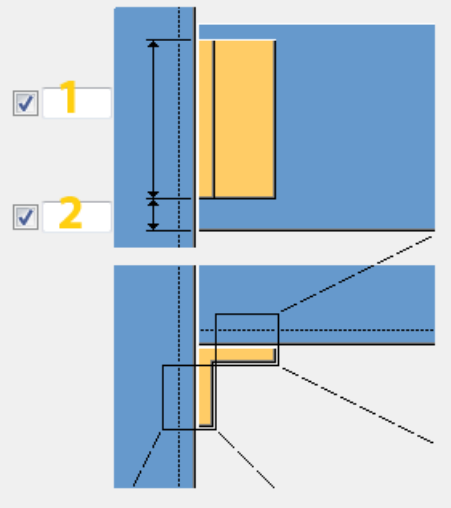


Part	Description	Default
L profile	Created only if brackets are created. Select the profile from the profile catalog.	`BLL80*80*10`

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the

Option	Description	Default
		Components settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

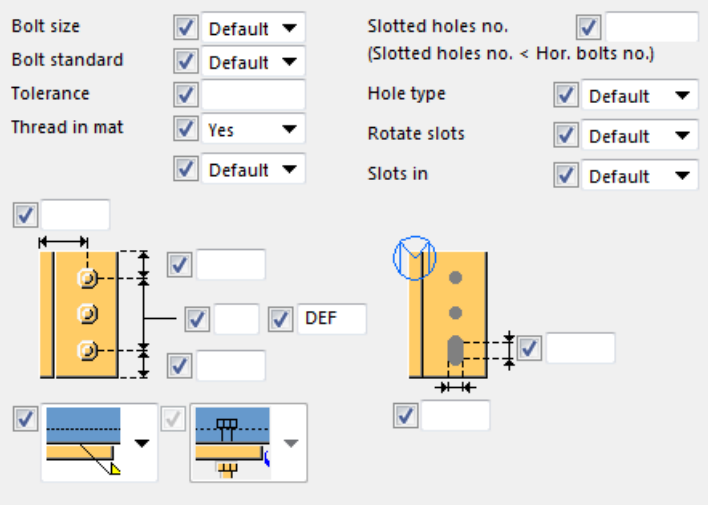
Bracket dimensions



	Description	Default
1	Define the height of the bracket.	Default height is calculated according to bracket-to-step or bracket-to-stringer bolt dimensions, depending on which one is bigger.
2	Define the dimension to offset the bracket from the Z pan edge.	25 mm

Bracket-to-step connection

Define the properties of the bolts that connect the brackets to the steps.

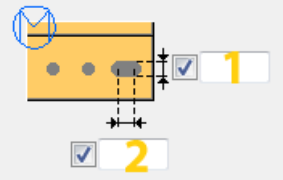


Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

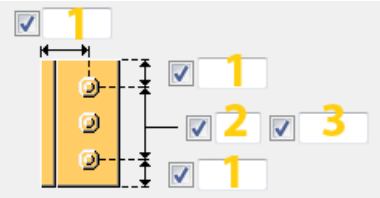
Slotted holes

You can define slotted, oversized, or tapped holes.



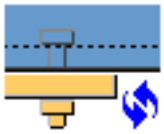
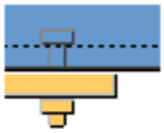
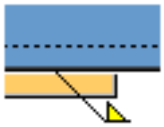
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Slotted holes no	Define the number of slotted holes.	
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt group dimensions

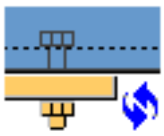
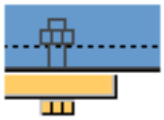
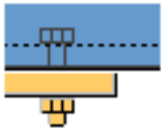


	Description	Default
1	Bolt edge distance.	`bolt diameter * 1.5`
2	Number of bolts.	`2`
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.	`100 mm`

Bracket-to-step connection type

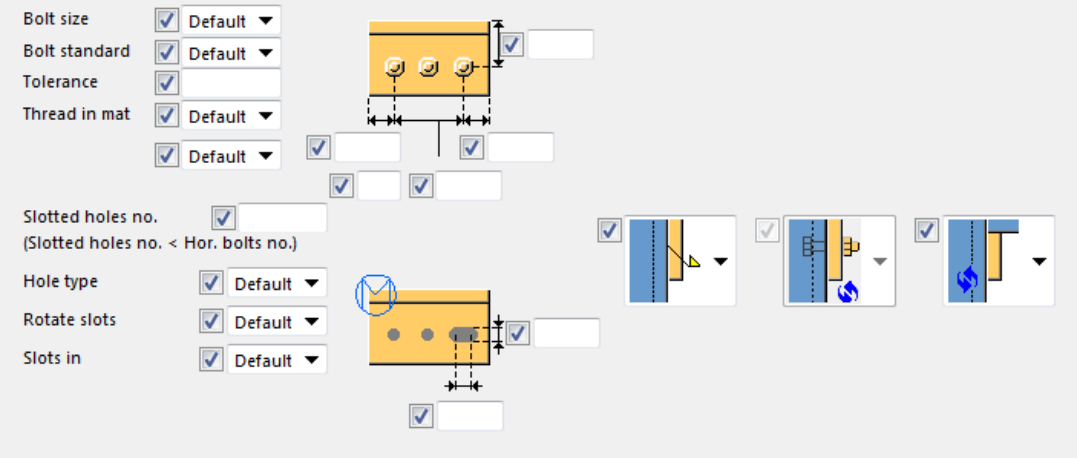
Option	Description
	Default.
	Bolted.
	Welded.
	No connection.
	No bracket angle leg. The bracket is a plate instead of an angle.
	Holes only.

Bolting direction

Option	Description
	Default.
	Bolted from the bracket to the step.
	Bolted from the step to the bracket.

Bracket-to-stringer connection

Define the properties of the bolts that connect the brackets to the stringers.

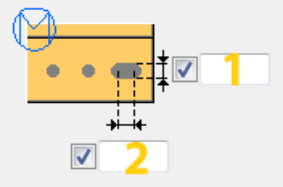


Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

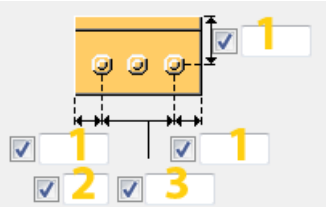
Slotted holes

You can define slotted, oversized, or tapped holes.



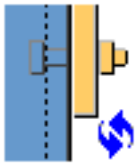
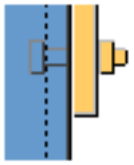
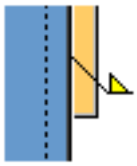
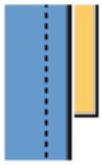
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Slotted holes no	Define the number of slotted holes.	
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt group dimensions

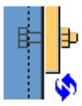
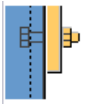
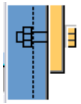


	Description	Default
1	Bolt edge distance.	`bolt diameter * 1.5`
2	Number of bolts.	`2`
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.	`100 mm`

Bracket-to-stringer connection type

Option	Description
	Default. Bolted. AutoDefaults can change this option.
	Bolted.
	Welded.
	No connection.

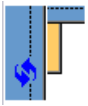
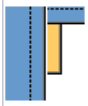
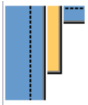
Bolting direction

Option	Description
	Default. Bolted from the bracket to the stringer. AutoDefaults can change this option.
	Bolted from the bracket to the stringer.
	Bolted from the stringer to the bracket.

Bracket position

Define the position of the plate-type bracket. The bracket can be positioned below the step, or between the step and the stringer.

You can define the bracket position if the bracket is a created as a plate instead of an angle.

Option	Description
	Default Below the step. AutoDefaults can change this option.
	Below the step.
	Between the stringer and step.

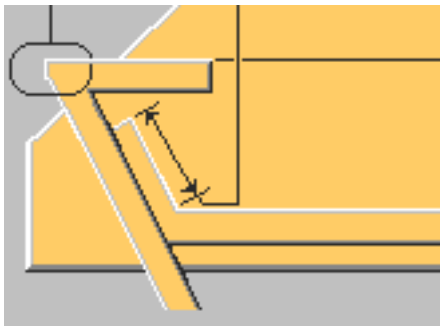
Bent plate bracket tab

Use the **Bent plate bracket** tab to control the dimensions and other properties of the bent plate bracket. You can use the bent plate bracket for connecting the Z pan steps to the stringers instead of horizontal or vertical brackets.

Preconditions

To create the bent plate bracket, set the following options:

- On the **Stair setup** tab, set **Bracket** to **Bent plate bracket**.
- On the **Z pan** tab, set **Front edge type** to the following option:



Bent plate profile

Part	Description	Default
Bent plate profile	Created if **Bracket** is set to **Bent plate bracket** on the **Stair setup** tab.	`BPL80*10`

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Create top step bent plate bracket

Select whether to create the bent bracket for the top step.

Option	Description
	Default Bent plate bracket is not created. AutoDefaults can change this option.
	Bent plate bracket is not created.
	Bent plate bracket is created.

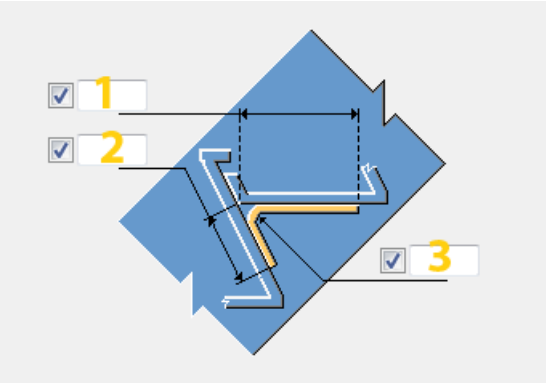
Create bottom step bent plate bracket

Select whether to create the bent bracket for the bottom step.

Option	Description
	Default Bent plate bracket is not created. AutoDefaults can change this option.

Option	Description
	Bent plate bracket is not created.
	Bent plate bracket is created.

Bent bracket dimensions



	Description	Default
1	Define the horizontal distance from the step corner to the bent plate edge.	`200 mm`
2	Define the vertical distance from the step corner to the bent plate edge.	`100 mm`
3	Define the radius of the round bending.	`1 / 3 * vertical dimension`

Welds

Click the link below to find out more:

Kickplate (S75)

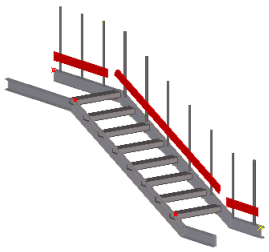
Kickplate (S75) creates one or more kickplates at the bottom of stanchions.

Objects created

- Kickplates

- Welds

Use for

Situation	Description
	Kickplates are connected to stanchions.

Before you start

Create stringers, beams, or slabs, and then stanchions using the **Stanchions (S76)** component, for example.

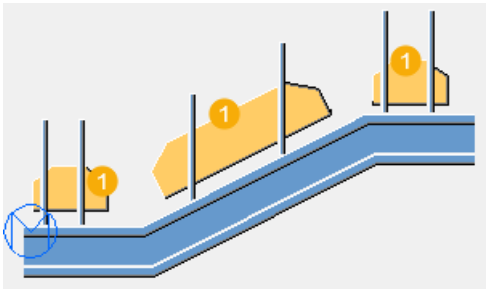
Selection order

1. Select the main parts (stringers/beams/slabs).
2. Select the first stanchion.
3. If **Stanchion selection** on the **Picture** tab is set to **Default** or **Use all**, and the stanchions have been created with **Stanchions (S76)**, click the middle mouse button to create the component.

   If the stanchions are modeled separately or **Stanchion selection** is set to **Use selected**, select the second stanchion, then third and so on, and click the middle mouse button.

Note that if you modify the component, all connected components will be deleted. For example, modifying stanchions deletes the connected kickplates and railings.

Part identification key



	Description
1	Kickplate

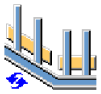
Picture tab

Use the **Picture** tab to define the kickplate dimensions, chamfers, and sides.

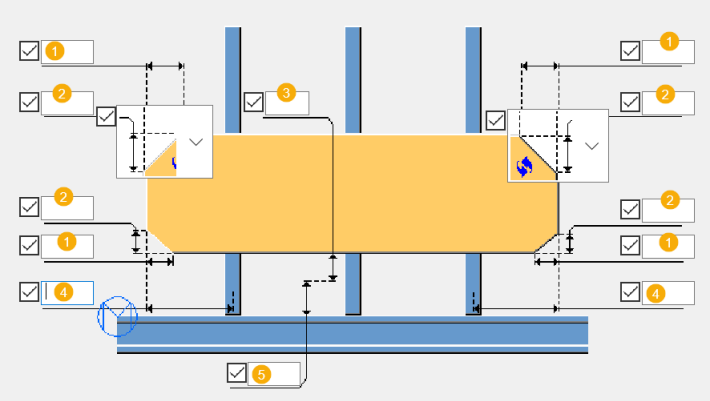
Part selection

Option	Description	Default
Stair parts selection	Define how the stringers or beams are considered: • **Use selected**: Only the selected stringers are considered for creating kickplates on. • **Use all**: If one or more of the selected stringers are part of a stair, then all the stringers in the same direction with the selected one, inside the stair, will be selected as well.	**Use all**
Stanchion selection	• **Use selected**: Only the selected stanchions are considered for creating kickplates on. • **Use all**: If one or more of the selected stanchions are part of a stanchions macro, then all the stanchions in the same direction with the selected one, inside the stair macro, will be selected as well.	**Use all**

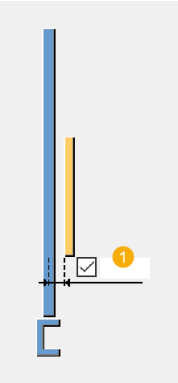
Kickplate on skewed stringer

	Default Kickplate is created on the skewed stringer. AutoDefaults can change this option.
	Kickplate is not created on the skewed stringer.
	Kickplate is created on the skewed stringer.

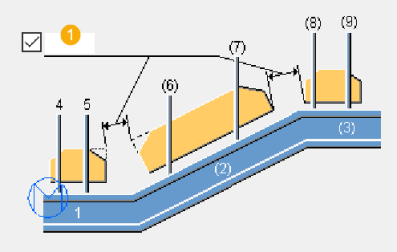
Dimensions



	Description	Default
1	Horizontal chamfer for the plate corner.	0 mm
2	Vertical chamfer for line chamfers and chamfer radius for round chamfers. If a kickplate is split in pieces, only the first and last piece will be chamfered.	0 mm
3	Kickplate height, referenced to the floor height.	200 mm
4	Kickplate extensions for the left or right sides.	150 mm
5	Floor height	0 mm



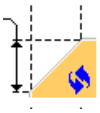
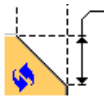
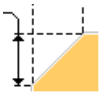
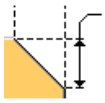
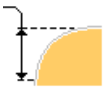
	Description	Default
1	Gap between the kickplates and the stanchions. The offset direction of the plates depends on which side of the stanchions the plates are created.	0 mm



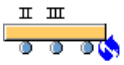
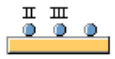
	Description	Default
1	Gap between two consecutive kickplates. The dimension is considered perpendicular on the bisecting lines between the two plates and is calculated from the closest corners between the points.	10 mm

Chamfer type

Chamfer type for the top vertical corners of the kickplate. If the plate is split in pieces, only the first and last piece are chamfered.

Option	Option	Description
		Default Line chamfer AutoDefaults can change this option.
		Line chamfer
		Round chamfer

Kickplate side

Option	Description
	Default Kickplates are created on the left side of the stanchions. AutoDefaults can change this option.
	Kickplates are created on the left side of the stanchions.
	Kickplates are created on the right side of the stanchions.

Option	Description
	Kickplates are centred on the stanchions line, and they are split in pieces between the stanchions.

Rotation

Select a suitable kickplate rotation option.

Options

Parameters tab

Use the **Parameters** tab to define the part properties, and kickplate cut and connection type.

Parts

Option	Description	Default
Kickplate	Select the profile from the profile catalog. The kickplate is created as a polygon plate if you do not select a profile.	
Profile max. length	Maximum length of the kickplate.	5000 mm

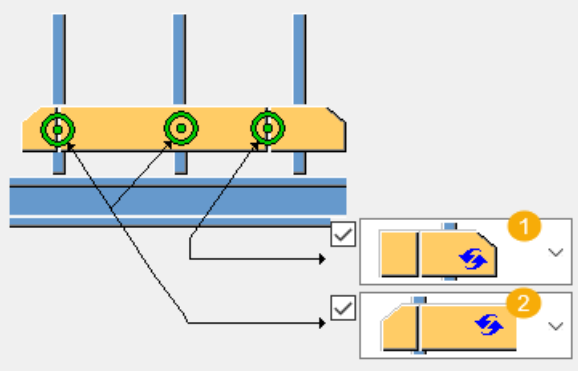
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
	where you can enter the assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

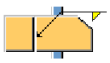
Kickplate cut type

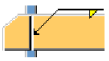
Option	Description
	Default No cut AutoDefaults can change this option.
	No cut
	Cut at maximum length The kickplate is split in pieces if its total length is more than defined in **Profile max length**.
	Cut at every stanchion The kickplate is split in pieces by cut planes defined by the stanchion centre lines.

Connection type



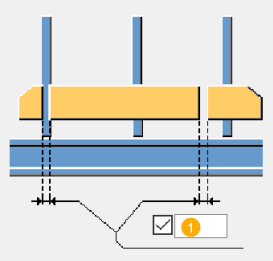
	Description
1	Connection type between one or two kickplate pieces and a stanchion (main part).
2	Connection type between two consecutive kickplate pieces.

Option	Description
	Default No connection AutoDefaults can change this option.
	No connection
	Welding
	Connection When you set the connection type to connection, a standard Tekla Structures connection is used as defined in **Connection name** and **Attribute file**.

Option	Description
	Default No connection AutoDefaults can change this option.
	No connection
	Welding

Option	Description
	Connection When you set the connection type to connection, a standard Tekla Structures connection is used as defined in **Connection name** and **Attribute file**.

Gap between plates



	Description
1	Gap between two consecutive pieces of the same kickplate.

Welds

Click the link below to find out more:

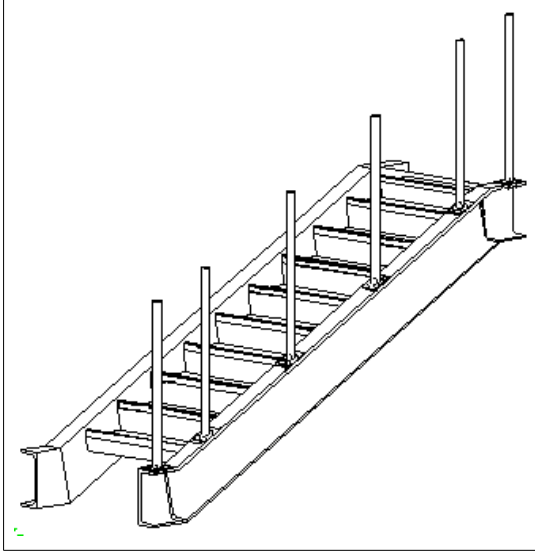
Stanchions (S76)

Stanchions (S76) creates stanchions on one or more horizontal or skew stair stringers. Stanchions can also be used on concrete slabs.

Objects created

- First stanchion
- Middle stanchions
- Last stanchion

Use for

Situation	Description
	Stanchions bolted on a stringer.

Limitations

NOTE If you modify **Stanchions (S76)**, all connected components are deleted. For example, modifying the stanchions deletes stairs, railings, and other components connected to the stanchions.

Before you start

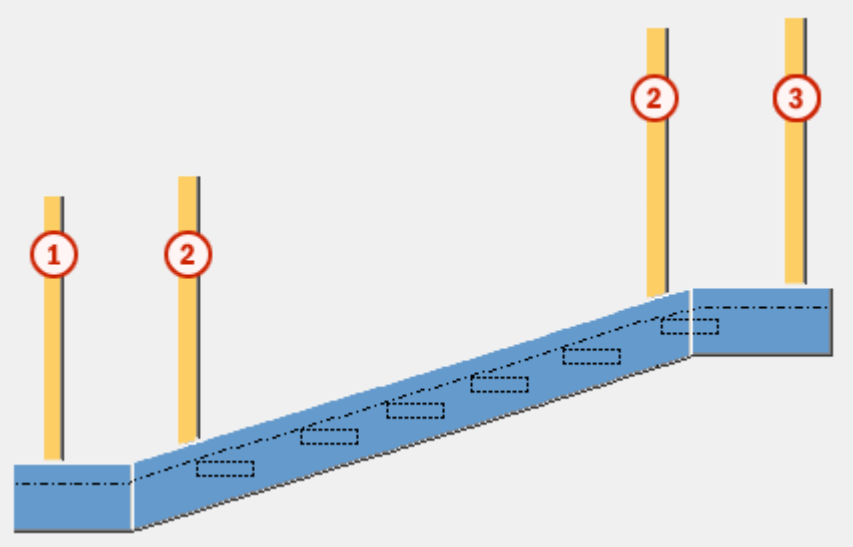
Create stringers, beams, or slabs or other parts to connect the stanchions.

Selection order

1. Pick the start point.
2. Pick the end point.
3. Select the lower landing (optional).
4. Select the stringer.
5. Select the upper landing (optional).

   Click the middle mouse button to create the component.

Part identification key



	Part
1	First stanchion
2	Middle stanchions
3	Last stanchion

Picture tab

Use the **Picture** tab to control the stanchion creation.

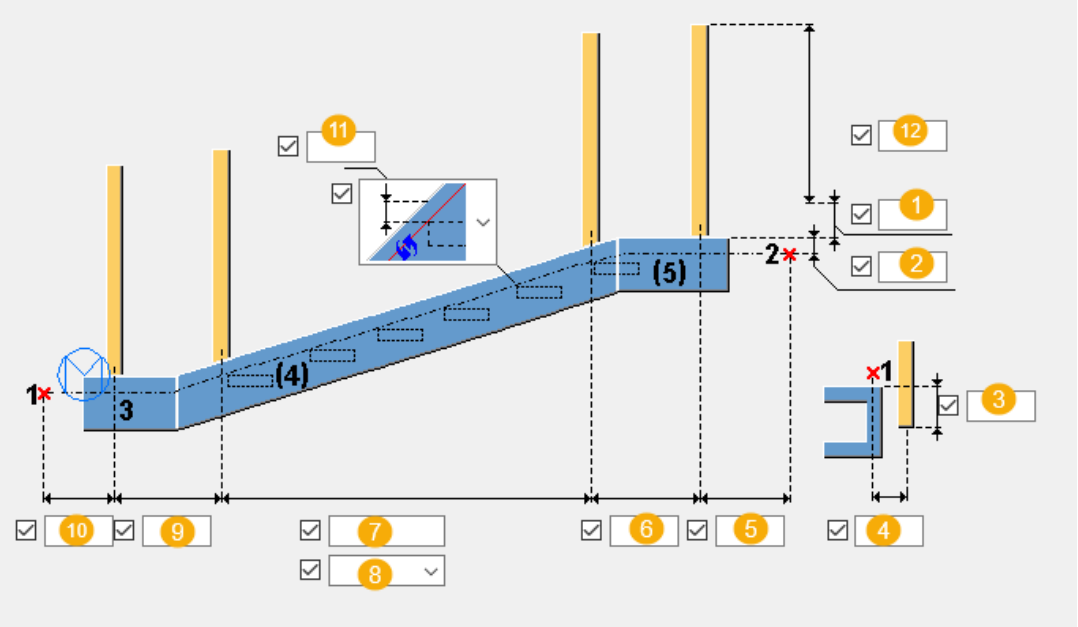
Stair parts selection

By default, the **Use all** option is used.

- **Use selected**: the stanchions are created only to the selected parts.
- **Use all**: if you have created one or more stringers using a stair component, select this option.

  All the stringers which are part of the stair component and have the same direction are selected.

Part positions

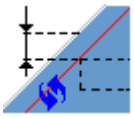
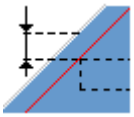
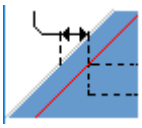
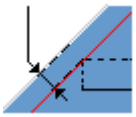


	Description	Default
1	Additional height to the stanchion.	
2	Height of the floor level.	1300 mm
3	Stanchion bottom point vertical offset.	0 mm
4	Stanchion bottom point horizontal offset.	0 mm
5	Distance between the second selected point and the last stanchion.	300 mm
6	Distance between the last two stanchions.	0 mm
7	Space between the stanchions. Use the value **8** to define the spacing type.	1000 mm
8	Stanchion spacing type. • Maximum The spacing value in the box **7** defines the maximum space between the stanchions. As many equally spaced middle stanchions as needed are created. The space between the stanchions is less or equal to the maximum distance. • Exact The space between the middle stanchions is exactly the spacing value in the box **7**. Enter the spacing values	Maximum

	Description	Default
	using the following formats: `6*950` or `1000 800 800 1000 800`.	
9	Distance between the first two stanchions.	`0 mm`
10	Distance between the first selected point and the first stanchion.	`300 mm`
11	Nosing dimension of the steps.	
12	Stanchion height from the floor level.	`1300 mm`

Nosing direction

Define the nosing direction.

Option	Description
	Default Vertical AutoDefaults can change this option.
	Vertical
	Horizontal
	Perpendicular

Parts tab

Use the **Parts** tab to control the stanchion profiles.

Stanchion profile

Option	Description
First Stanchion	Define the first stanchion profile by selecting it from the profile catalog. First stanchion is always created. The default name is `STANCHION`.

Option	Description
Middle Stanchion	Define the middle stanchion profile by selecting it from the profile catalog. Middle stanchion is always created. The default name is `STANCHION`.
Last Stanchion	Define the last stanchion profile by selecting it from the profile catalog. Last stanchion is always created. The default name is `STANCHION`.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Stanchion profile rotation

Define the stanchion profile rotation and position.

Option	Description	Default
Profile rotation	Stanchion profile rotation and position.	Rotation = `Front` Position = `Middle`
Rotation value	Stanchion profile rotation angle (in	Rotation angle = `0` Offset = `0 mm`

Option	Description	Default
	degrees) and the plane offset.	

Stanchion profile rotation type

Define the rotation combinations for consecutive stanchions.

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2
	Type 3
	Type 4
	Type 5

Parameters tab

Use the **Parameters** tab to control how the stanchions are connected to the stairs.

Connection properties

Option	Description
Connection setting type	Define whether pre-defined or custom settings are used for connecting the stanchions to the stairs. By default, the pre-defined settings are used.
Stanchion connection type	Connection type for stanchions. By default, the weld default connection type is used. This field is active only if you have set the **Connection setting type** to **Pre-defined settings**.
Connection number	Define a connection that connects stanchions to the stairs by selecting it from the component catalog.

Option	Description
Connection properties	Select an attribute file for the connection.
Direction	Connection direction.
Stanchion product name	Enter the product name.

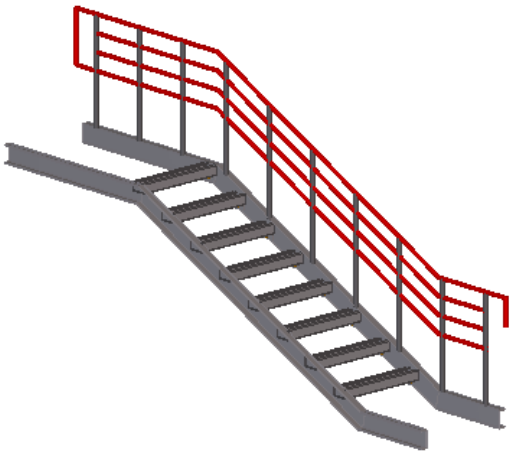
Railings (S77)

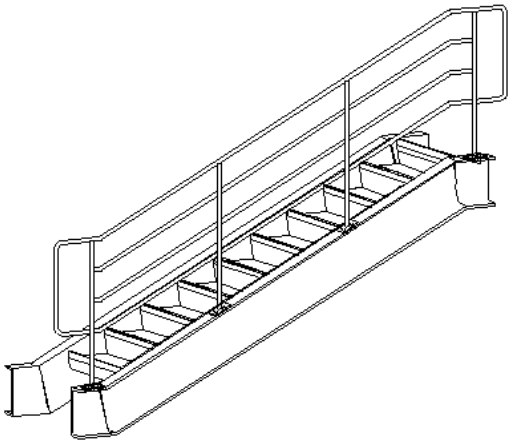
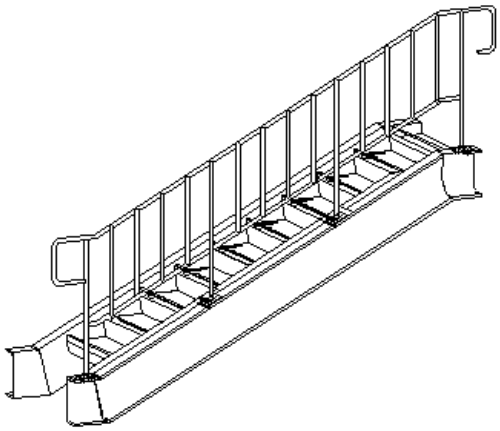
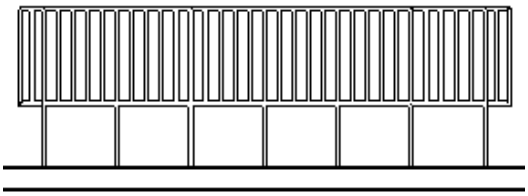
Railings (S77) creates railings connected to two or more stanchions. The railings can be handrails in stairs or guardrails in a horizontal beam or slab.

Objects created

- Top rail
- Middle rails (horizontal or vertical)
- Bottom rail
- Panels
- Bends

Use for

Situation	Description
	Horizontal railings

Situation	Description
	Horizontal railings connected to four stanchions
	Vertical railings connected to four stanchions
	Guardrail with vertical railings

Before you start

Create the stanchions using, for example, **Stanchions (S76)**.

Selection order

1. Select the first stanchion.
2. If the stanchions are created with **Stanchions (S76)**, and on the **Picture** tab the **Stanchion selection** is set to **Use all** or **Default**, the railings are created when you click the middle mouse button.

OR

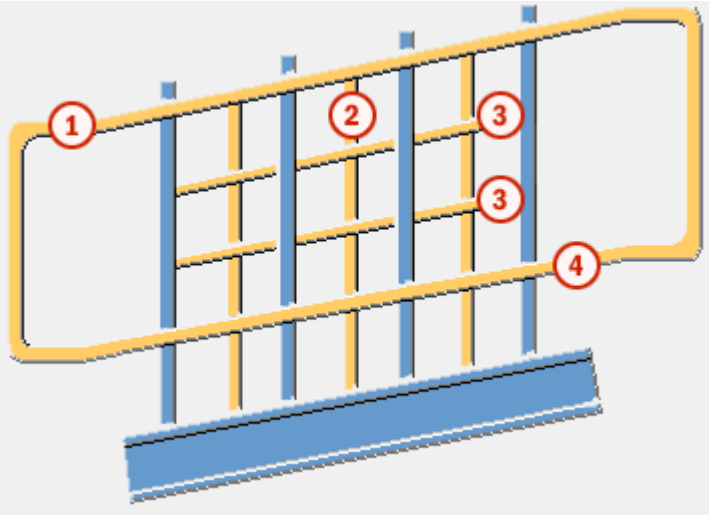
1. Select the first stanchion.

2. If the stanchions are created separately, or on the **Picture** tab the **Stanchion selection** is set to **Use selected**, select the second, third, and so on, stanchion.

   The order in which the stanchions are selected defines the positioning of the railings. The two first selected stanchions define the left and right directions of the railings.

3. Click the middle mouse button to create the railings.

Part identification key



	Part
1	Top rail
2	Vertical rails
3	Middle rails
4	Bottom rail

Picture tab

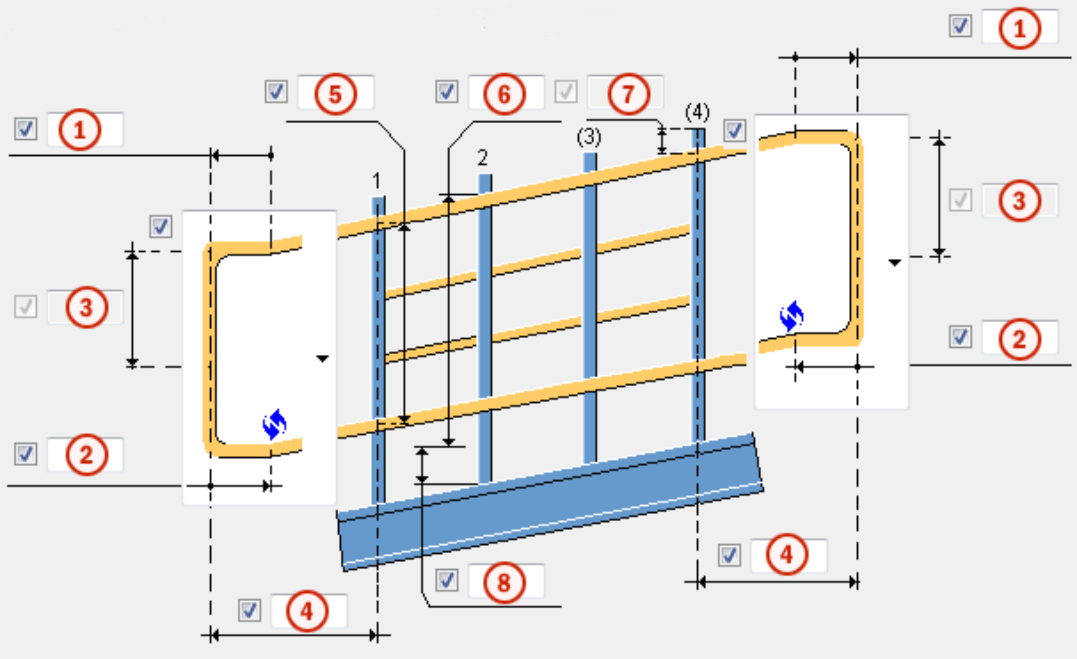
Use the **Picture** tab to control the railings and the closures.

Stanchion selection

By default, the **Use all** option is used.

- **Use selected**: the railings are created only to the selected stanchions.
- **Use all**: select this option when the stanchions are created with **Stanchions (S76)**.

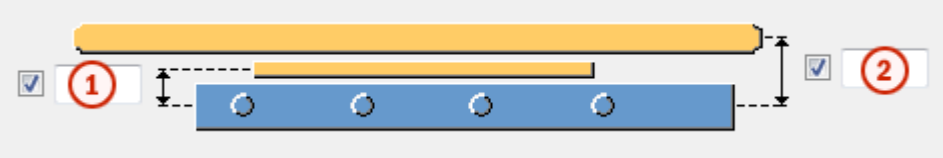
Part positions



	Description	Default
1	Horizontal length of the closure in the top rail and where the bending is created in the closure. When you use the closure type 4 and enter a negative value in this box, the top and bottom rails are shortened.	`150 mm`
2	Horizontal length of the closure in the bottom rail. When you use the closure type 4 and the value in the **1** box is `0`, you can shorten the bottom rails by entering a positive value in this box.	`150 mm`
3	Vertical length of the closure. Available for closure types 2 and 3.	`half the value of` **5**
4	Full horizontal length of the closure from the first or the last stanchion.	`300 mm`
5	Distance between the top and the bottom rails. If the middle rails are positioned evenly between the top and bottom rails, the spacing between them is adjusted if the distance between the top and the bottom rails is modified.	`800 mm`

	Description	Default
6	Height of the railing from the floor level. If you modify the floor level, the height of the railing is modified as well.	`defined by the height of the stanchion`
7	Length of the stanchion extension over the railing. Use this option if the **Top rail to stanchion fitting** is set to **Stanchion partcut** on the **Parameters** tab.	`0 mm`
8	Floor level from the bottom of the stanchion. If you modify the floor level, the height of the railing is modified as well.	`0 mm`

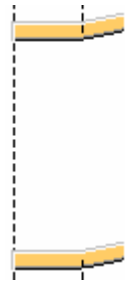
Railing offset



	Description
1	Vertical distance between the middle rails and the stanchion center.
2	Vertical distance between the top or the bottom rail and the stanchion center.

Left and right closure

Option		Description
		Default Type 1 AutoDefaults can change this option.
		Type 1

Option		Description
		Type 2
		Type 3
		Type 4

Parameters tab

Use the **Parameters** tab to control the cuts, fitting, and gaps between the railing parts.

Continuity profile

Option	Description
Top Continuity profile	Define the continuity profile by selecting it from the profile catalog. The top continuity profile is created if the length of the continuous profile is more than `0` and **Top rail parts gap** is set to **Yes**. The default name is `RAIL`.
Middle Continuity profile	Define the continuity profile by selecting it from the profile catalog. The middle continuity profile is created if the length of the continuous profile is more than `0` and **Midd. rail parts gap** is set to **Yes**. The default name is `RAIL`.

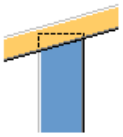
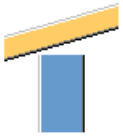
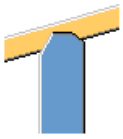
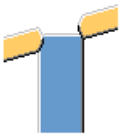
Option	Description
Bottom Continuity profile	Define the continuity profile by selecting it from the profile catalog. The bottom continuity profile is created if the length of the continuous profile is more than `0` and **Bott. rail parts gap** is set to **Yes**. The default name is `RAIL`.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Rail length

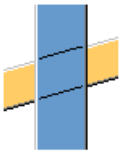
Option	Description	Default
Top rail max. length	Maximum length of the top rail.	`3000 mm`
Middle rail max. length	Maximum length of the middle rail.	`3000 mm`
Bottom rail max. length	Maximum length of the bottom rail.	`3000 mm`

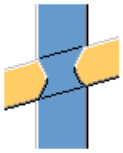
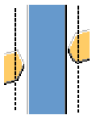
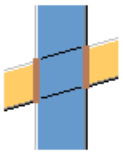
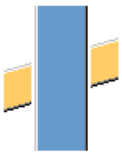
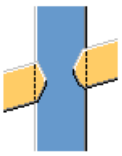
Rail fittings

Option	Description
Top rail to stanchion fitting	Select the fitting type between the top rail and the stanchions. Define the cut tolerance and the fitting distance using the **Top stanchion cut tolerance** and **Top rail to stanchion fitting distance** options. The parts are first cut and then fitted. By default, no fitting is created. •  No fitting is created. • Stanchions are fitted to rail. •  Stanchions are cut by a horizontal plane. •  Stanchions are cut by the rails. •  Rails are cut by the stanchions.
Top rail to stanchion fitting distance	Fitting distance between the top rail center line and the stanchion.

Option	Description
Top stanchion cut tolerance	Cut tolerance between the top rail and the stanchions.

Option	Description
Top rail at end stanchions	Select the fitting type between the top rail and the first or last stanchion if the start or end closure is not created. • Corner is not fitted. This is the default option. • Corner is fitted on both sides and a bent part is created. • Corner is not fitted. • Corner is fitted in the middle.

Option	Description
Other rail to stanchion fitting	Select the fitting type between the middle or bottom rail and the stanchions. Define the cut tolerance and the fitting distance using the **Other stanchion cut tolerance** and **Other rail to stanchion fitting distance** options. The parts are first cut and then fitted. By default, no cuts are created. •  No cut is created.

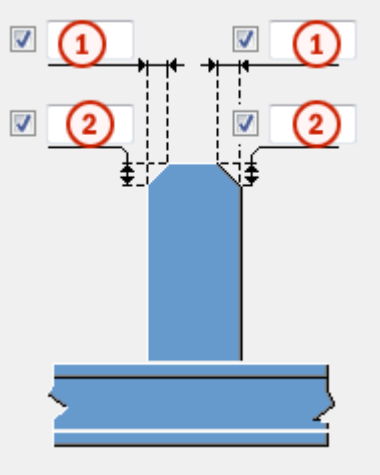
Option	Description
	•  Stanchions are cut by the rails. •  Rails are cut by the stanchions. •  Stanchions are cut by the rails through a slotted hole. •  Rails are fitted to the stanchions. •  Rails are cut by the stanchions without any tolerance.
Other rail to stanchion fitting distance	Fitting distance between the middle or bottom rail center line and the stanchion.
Other stanchion cut tolerance	Part cut tolerance between the middle or the bottom rail and the stanchions.

Rail cuts

Option	Description
Top rail cuts	Define how the top rail is cut. By default, the **At max stanchion** option is used.

Option	Description
	• **At max stanchion** Starting from the first stanchion, without the closures, the **Top rail max. length** is measured along the rail and the cut is made at the last stanchion included in the measured distance. • **At max length** Starting from the first point of the rail, including the closure, the **Top rail max. length** is measured along the rail and the cut is made at the maximum length. • **Every stanchion** A cut is made to the rail at each stanchion. • **At max length no closure** Starting from the first stanchion, without the closures, the **Top rail max. length** is measured along the rail and the cut is made at the maximum length.
Middle rail cuts	Define how the middle rail is cut. By default, the **At max stanchion** option is used.
Bottom rail cuts	Define how the bottom rail is cut. By default, the **At max stanchion** option is used.

Chamfer dimensions

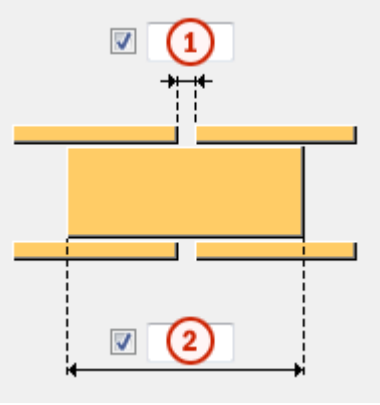


	Description
1	Horizontal chamfer dimension for the stanchions.
2	Vertical chamfer dimension for the stanchions.

Gap

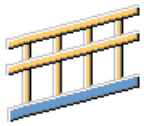
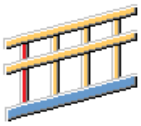
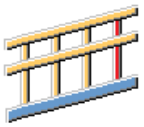
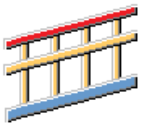
Option	Description
Top rail parts gap **Midd. rail parts gap** **Bott. rail parts gap**	Select whether a gap is created between the rail parts. By default, no gap is created.

Gap between parts



	Description
1	Size of the gap between two consecutive rail parts.
2	Length of the continuous profile.

Assembly

Option	Description
Create assembly	Select which part of the railing is the main part of the assembly. By default, no assembly is created. •  No assembly is created. This is also the default option. •  The first stanchion is the main part of the assembly. •  The last stanchion is the main part of the assembly. •  The first rail part after the first stanchion is the main part of the assembly. If you have created continuous profiles for all the rails, multiple assemblies are created.

Rails tab

Use the **Rails** tab to control the profiles, bending cuts, and connections for the top and the bottom rails.

Rail and closure profiles

Option	Description
Top rail profile	Define the top rail profile by selecting it from the profile catalog. Top rail is always created.
Bottom rail profile	Define the bottom rail profile by selecting it from the profile catalog. Bottom rail is always created.
Start closure	Define the start closure by selecting it from the profile catalog. By default, the **Start closure** profile is the same as the **Top rail profile**.
End closure	Define the end closure by selecting it from the profile catalog. By default, the **End closure** profile is the same as the **Top rail profile**.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Closure bending cuts

Option	Description
	Default No cut AutoDefaults can change this option.
	No cut No cut is created near the corner.
	Both side cuts The rail corner is cut at the left and the right side. The dimensions for the cut are defined on the **Bends** tab.
	Middle corner cut The rail corner is cut in the middle.
	Mitre corner cut The rail corner is cut by a plane on the bisecting line.
	Left side cut The rail corner is cut at the left side. The dimensions for the cut are defined on the **Bends** tab.
	Right side cut The rail corner is cut at the right side. The dimensions for the cut are defined on the **Bends** tab.

Cut position of the rails

Define the cut position for the top and the bottom rails and for the closure types 1 and 4.

Option	Description
	Default Closure near side cut AutoDefaults can change this option.

Option	Description
	Closure near side cut The rail passes through the first or the last stanchion and is cut at its surface. The closure is cut at the near side surface of the stanchion, without intersecting it.
	Middle stanchion cut The rail passes through the first or the last stanchion and is cut at its middle line. The closure is cut at the middle line of the stanchion, intersecting it.
	Closure far side cut The rail is cut at the surface of the stanchion, without intersecting it. The closure is cut at the other far side surface of the stanchion, intersecting it.

Cut position for the other bending

Define the cut position for the other bendings than the corners of the closures.

Option	Description
	Default No cut is created. AutoDefaults can change this option.
	No cut is created.
	Both side cuts The rail bending is cut at the left and the right side. The dimensions for the cut are defined on the **Bends** tab.
	Middle corner cut The rail bending is cut in the middle.

Option	Description
	Mitre corner cut The rail bending is cut by a plane on the bisecting line.

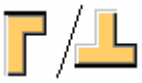
Top rail position / Bottom rail position

Define the horizontal railing position.

By default, the **Middle** option is used.

Top rail rotation / Bottom rail rotation

Define the profile rotation for all horizontal rails.

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2
	Type 3
	Type 4
	Type 5
	Type 6
	Type 7
	Type 8

Top rail and bottom rail connection properties

Option	Description
Top rail connection, **Bottom rail connection**	Select whether to use a system or a custom component to connect the

Option	Description
	top and bottom rails to the stanchions.
Component	Select a system or custom component from the **Applications & components** catalog.
Attribute file	Select an attribute file for the component.
Up direction	Select the up direction.
Rotation	Select the rotation for the selected component.
Rotation value	Enter the rotation value.

Middle Rails tab

Use the **Middle Rails** tab to control the horizontal middle rails.

Rail profile

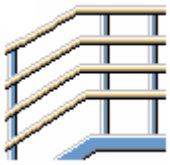
Option	Description
1 **2** **3** **4**	Define the middle rail profile by selecting it from the profile catalog. By default, the size of middle rail profile is the same as the size of the top rail profile. The default name is `MIDDLE RAIL`.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Option	Description	Default
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Middle rail type

Option	Description
Middle rail profiles	Number of rails and the rail profile used for middle rails. For example, `3*2` creates three horizontal middle rails, with the profile defined in profile **2**. For example, `2*2 1` creates two horizontal middle rails, with the profile defined in profile **2** and one or more rails (depending of the number of rails needed), with the profile defined in profile **1**. For example, `3` creates horizontal middle rails with the profile defined in profile **3**. The number of rails is defined by the number set for the horizontal or vertical middle rails. By default, only rails with profile from profile **1** are created.

Option	Description
	Default Creates horizontal middle rails. AutoDefaults can change this option.
	Creates horizontal middle rails.
	Creates vertical middle rails (pickets) defined on the **Vertical Rails** tab.

Option	Description
	Creates panels.

Middle rail position

Define the horizontal railing position.

By default, the **Middle** option is used.

Middle rail rotation

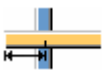
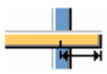
Define the profile rotation for all horizontal rails.

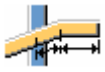
Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2
	Type 3
	Type 4
	Type 5
	Type 6
	Type 7
	Type 8

Cut position for the bending

Option	Description
	Default No cut AutoDefaults can change this option.
	No cut No cut is created.
	Both side cuts The rail bending is cut at the left and right side. The dimensions for the cut are defined on the **Bends** tab.
	Middle corner cut The rail bending is cut in the middle.
	Mitre corner cut The rail bending is cut by a plane on the bisecting line.

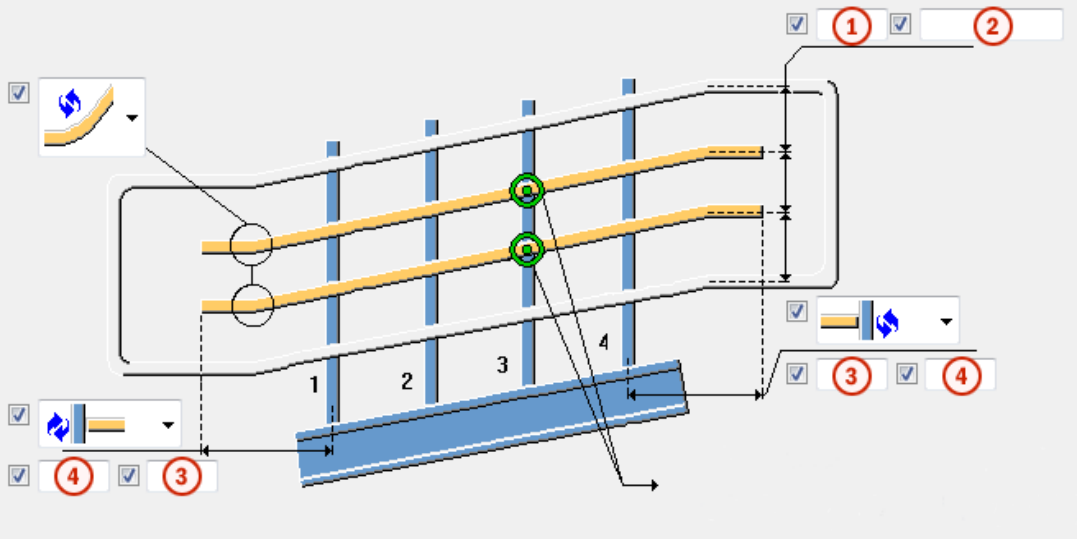
Horizontal middle rail extension

Option		Description
		Default No extension AutoDefaults can change this option.
		No extension The middle rail ends at the inside stanchion surface, without intersecting the stanchion.
		Pass through stanchion The middle rail ends at the outside stanchion surface, intersecting the stanchion.
		Horizontal extension The middle rails are extended inside the

Option		Description
		closure by a horizontal value. If the railing is skew, the extension is bent according to the closure dimensions. The rail can be shortened by entering a negative value in the box **3**.
		Horizontal and skew extension The middle rails are extended by two horizontal dimensions. **3** is for the horizontal rail length and **4** is for the skew rail length.

Closure bend cuts

This section is not active if you have set the middle rail type to vertical middle rails.



	Description
1	Number of horizontal middle rails.
2	Spacing between the horizontal middle rails.

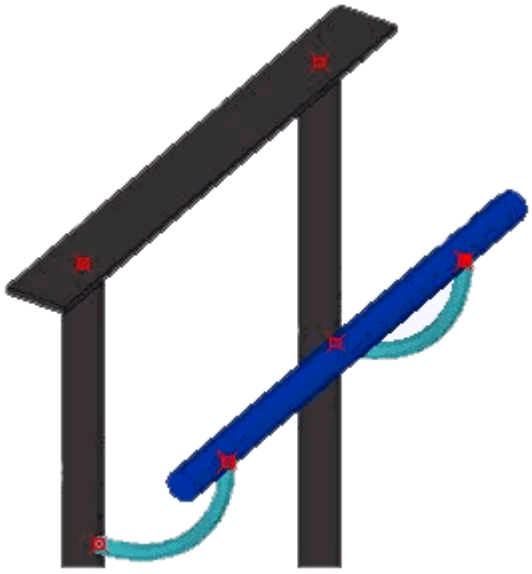
	Description
3	Length of the middle rail horizontal extension inside the closure. The rail can be shortened by entering a negative value. Use this box when the horizontal middle rail extension is set to horizontal extension.
4	Length of the middle rail horizontal and skew extension. The middle rails are extended by two horizontal dimensions. **3** is for the horizontal rail length and **4** is for the skew rail length. Use this box when the horizontal middle rail extension is set to horizontal and skew extension.

Middle rail connection properties

Option	Description
Middle rail connection	Select whether to use a system or custom component to connect the middle rails to the stanchions.
Component	Select a system or custom component from the **Applications & components** catalog.
Attribute file	Select an attribute file for the component.
Up direction	Select the up direction.
Rotation	Select the rotation for the selected component.
Rotation value	Enter the rotation value.
Connect to	Select whether one rail or two rails are connected to stanchions.

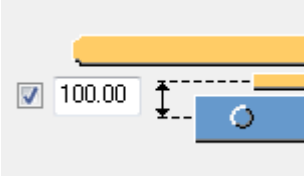
Grab rail

Grab rail can be a single middle rail that is vertically and horizontally offset from the top rail. You can use a system or a custom component to connect the rail to stanchions. You can also select the end of the grab rail. If the grab rail comes from a skew stringer, you can extend the grab rail with skew and horizontal dimensions.

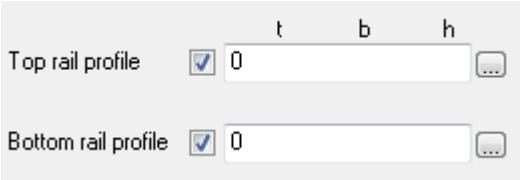


To create a grab rail:

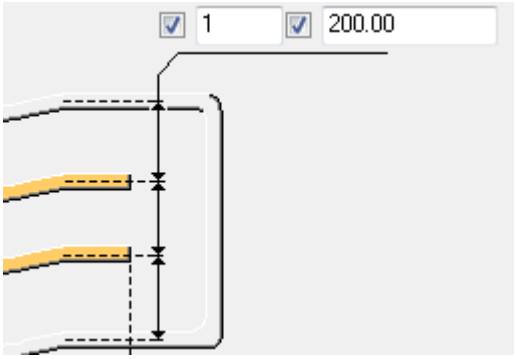
1. On the **Picture** tab, offset the middle rails from the stanchions.



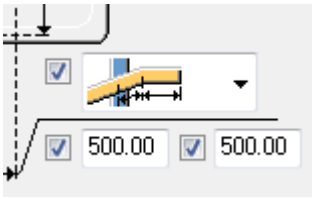
2. On the **Rails** tab, set the profile for top and bottom rails to `0` so that those rails are not created.



3. On the **Middle Rails** tab, set the number of horizontal rails to `1` and adjust the spacing for the handrail.

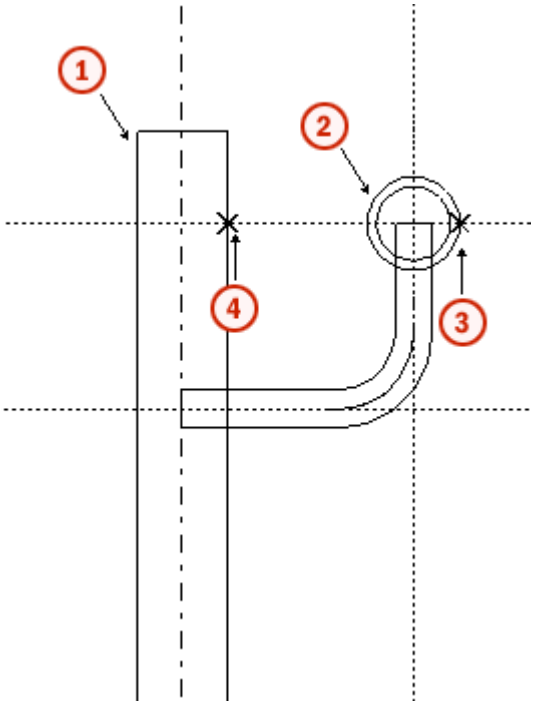


4. If you want to extend the end of the grab rail and have a horizontal part, select the horizontal and skew extension option on the **Middle Rails** tab, and enter the skew and horizontal distances.



5. Create the custom seam.

   Pick the points on the outer extension of the main and secondary parts.



	Description
1	Main part
2	Secondary part
3	First picked point
4	Second picked point

6. Save the standard properties for the custom seam.
7. On the **Middle Rails** tab, use the custom seam name and properties and set the component direction.

Vertical Rails tab

Use the **Vertical Rails** tab to control the vertical rails (pickets). The options on the **Vertical Rails** tab are active only if you have set the middle rail type to vertical middle rails on the **Middle Rails** tab.

Pickets

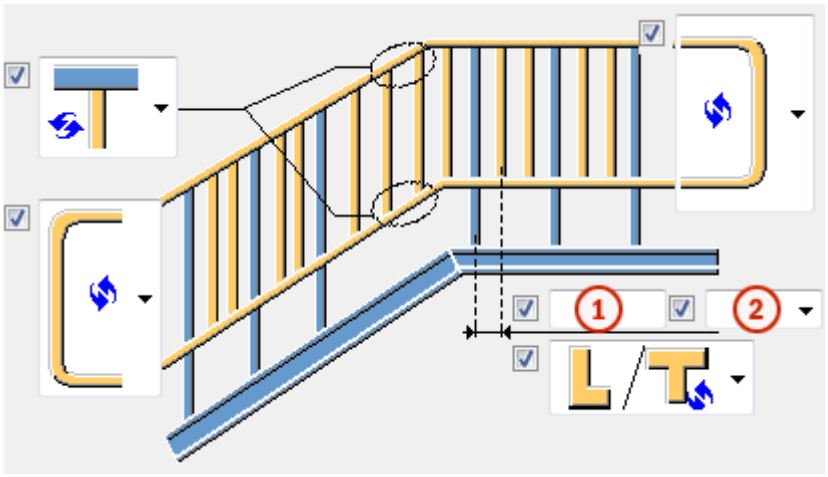
Option		Description
		Default Pickets are not created inside the closure. AutoDefaults can change this option.
		Pickets are not created inside the closure.
		Pickets are created inside the closure when you use the closure type 1 or 4.

Connection type between the pickets and the top and bottom rails

Option	Description
	Default None AutoDefaults can change this option.
	None The pickets are created from the top rail center to the bottom rail center.
	Fitted The pickets are fitted at the creation points by a plane that follows the slope of the horizontal rails.
	Part cut The pickets are cut on both sides by the horizontal rail profiles.

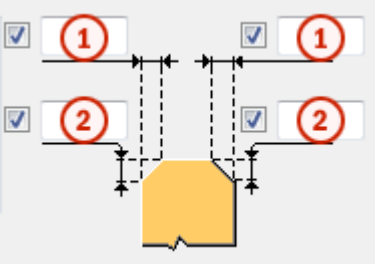
Option	Description
	Connection The pickets are connected to the top and bottom rail with a system connection.

Picket spacing



Option	Description
1	Picket spacing between two consecutive stanchions.
2	Spacing type. By default, the equal spacing option is used. • Exact The space between two consecutive pickets is exactly the spacing value **1**. As many pickets as possible are positioned between two consecutive stanchions. The pickets are positioned centered between the stanchions so that the space between the first stanchion and the first pickets and the second stanchion and the last pickets is equal. • Equal The spacing value **1** defines the maximum spacing. The pickets are placed evenly between the stanchions.

Chamfer dimensions



	Description
1	Horizontal chamfer dimension for the pickets.
2	Vertical chamfer dimension for the pickets.

Profile rotation

Define the profile rotation for all vertical rails.

Option	Description
	Default
	Type 1
	Type 2
	Type 3
	Type 4
	Type 5
	Type 6
	Type 7
	Type 8

Connection properties

Option	Description
Vertical rail top joint	Define the number of the component that connects the pickets to the top rail by selecting it from the component catalog. You cannot use a custom component. Select also an attribute file for the component.
Vertical rail bottom joint	Define the number of the connection that connects the pickets to the bottom rail by selecting it from the component catalog. You cannot use a custom component. Select also an attribute file for the component.

Panels tab

Use the **Panels** tab to create panels between the stanchions. No other vertical or horizontal rails are created when the panels are created. The options on the **Panels** tab are active only if you have set the middle rail type to panels on the **Middle Rails** tab. You can use custom seams to create the middle panels and custom details to create the closure panels.

Panel properties

Option	Description
Part name/Joint no.	Define the number of the custom component used in panel creation by selecting it from the component catalog.
Attribute file	Select an attribute file for the custom component.
Seam dir.	Define the seam direction.

Panel type

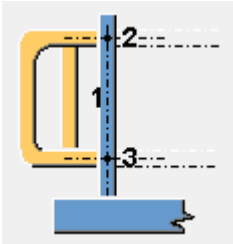
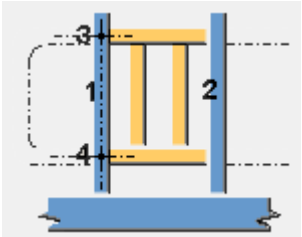
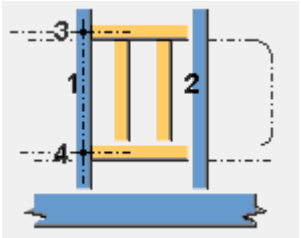
For custom details the numbers in the images correspond to the following parts:

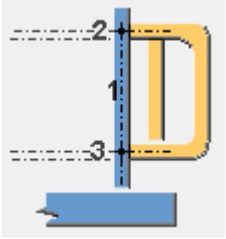
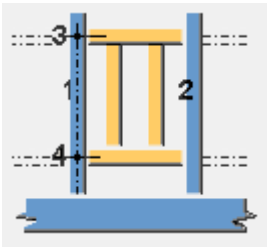
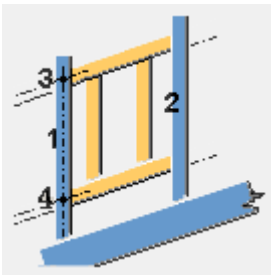
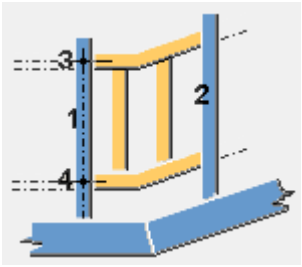
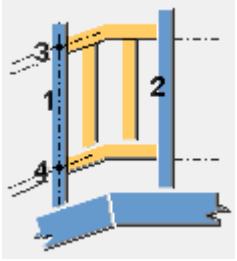
Number	Part
1	First stanchion, which is the main part of the detail.

Number	Part
2	Point at the top of the first stanchion, which is the first input point of the detail.
3	Point at the bottom of the first stanchion, which is the second input point of the detail.

For custom seams the numbers in the images correspond to the following parts:

Number	Part
1	First stanchion, which is the main part of the seam.
2	Second stanchion, which is the secondary part of the seam.
3	Point at the top of the first stanchion, which is the first input point of the seam.
4	Point at the bottom of the first stanchion, which is the second input point of the seam.

Option	Description
	Left closure panel Use a custom detail to create the panel. The panel connects to one stanchion only.
	First panel Use a custom seam to create the panel. The panel can be of different size than the rest of the panels.
	Last panel Use a custom seam to create the panel. The panel can be of different size than the rest of the panels.

Option	Description
	Right closure panel Use a custom detail to create the panel. The panel connects to one stanchion only.
	Regular horizontal panel Use a custom seam to create the panel.
	Regular skew panel Use a custom seam to create the panel.
	Horizontal panel, skew junction (bottom) Use a custom seam to create the panel.
	Horizontal panel, skew junction (top) Use a custom seam to create the panel.

Bends tab

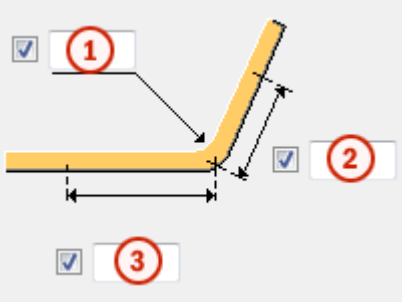
Use the **Bends** tab to set the bending cut position on the rails and the profiles for the bent rails parts.

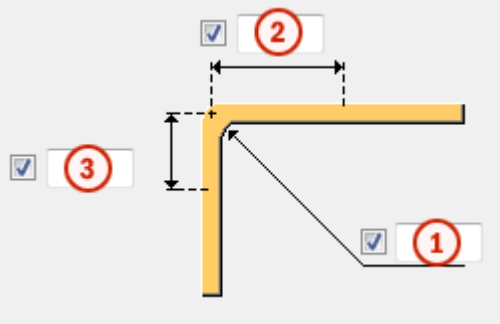
Bend profile

Option	Description
Bend	Define the bend rail profile by selecting it from the profile catalog. Created only if you have set the closure bend cuts to both side cut, left side cut, or right side cut on the **Rails** tab.
90 degree	Define the rail profile by selecting it from the profile catalog. Created only if you have set the cut position for the bending to both side cut on the **Rails** or **Middle Rails** tab.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Bend dimensions





Option	Description	Default
1	Bending radius for the rail parts.	50 mm
2	Cut distance along the rails on the right side of the bending.	75 mm
3	Cut distance along the rails on the left side of the bending.	75 mm

Welds

Click the link below to find out more:

Stairs (S82)

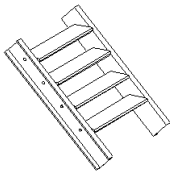
Stairs (S82) creates straight stairs with optional upper and lower landings. The stairs consist of stringers, possible landings and the actual steps. The stairs are created between two picked positions that define the upper and lower positions of the sloping stringers, taking into account the flooring thickness. The lower position defines the top level of the lowest step and the upper position the top level of the topmost step.

Objects created

- Stringers
- Steps

- Landings (optional)
- Welds

Use for

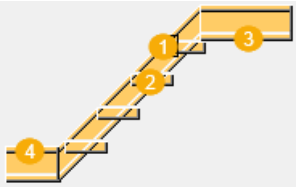
Situation	Description
	Stairs created with stringers and steps

Selection order

1. Pick the first position.
2. Pick the second position.

   The stairs are created automatically when you pick the second position.

Part identification key

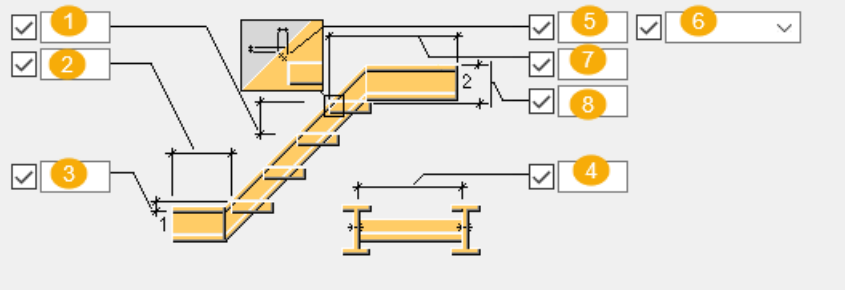


	Description
1	Stringer
2	Step
3	Top landing
4	Bottom landing

Picture tab

Use the **Picture** tab to define the stair dimensions.

Stair dimensions



	Description	Default
1	Maximum height between the steps.	230 mm The default value for the step height is calculated as follows: (Z*220)/(Z+220) mm, where Z is the height of the stairs.
2	Bottom landing length You need to define the length to create the landing.	Bottom landing beam is not created.
3	Bottom flooring thickness. Vertical distance from the lower picked point to the stringer top level.	30 mm
4	The distance the steps are inset between the stringers.	20 mm
5	Top flooring thickness Vertical distance from the upper picked point to the stringer top level.	30 mm
6	Vertical distance orientation	
7	Width of the stairs (horizontal length of the steps)	1000 mm
8	Top landing length You need to define the length to create the landing.	Top landing beam is not created.

Parts tab

Use the **Parts** tab to define the stringer profile properties.

Parts

Part	Description	Default
Stringer profile	Select the profile for stringers and landing beams from the profile catalog.	U200

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
	assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

Use the **Parameters** tab to control the stringer rotation, mirroring, position in plane, step profile, and top and bottom step creation.

Option	Description	Default
Stringer rotation	Define the stringer rotation around its axis on the work plane.	**Top**
Mirroring	Select whether the stringer is mirrored.	
Position in plane	Define the stair position on the work plane.	**Right**
Offset	Define the beam offset for the selected position in plane.	`0`
Step profile	Select the step profile. The step types are more precisely defined in the `steps.dat` file. The file defines the form of the step, and the place and size of the holes in the stringers. Each line in the file describes one step type. The actual step is created as a contour plate.	
Create top step	Select whether the top step is created.	**Yes**, top step is created.
Create bottom step	Select whether the bottom step is created.	**Yes**, bottom step is created.

Option	Description	Default
Create assembly	Select whether to create an assembly that includes all the parts of the stairs or stringers. With **All**, the steps are welded to the stringers with invisible weldings and an assembly drawing of the stairs can be created.	**No**
Bolt tolerance	Define the bolt hole tolerance for catalogue steps.	

Welds

Click the link below to find out more:

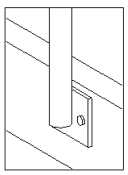
Stanchion side plate (83)

Stanchion side plate (83) connects a stanchion to the side of a stringer with a connection plate.

Objects created

- Connection plate
- Bolts
- Welds

Use for

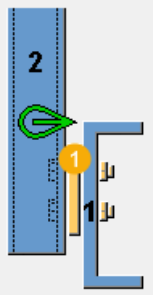
Situation	Description
	Stanchion is connected to the side of a stringer.

Selection order

1. Select the main part.
2. Select the secondary part.

   The connection is created automatically when the secondary part is selected.

Part identification key

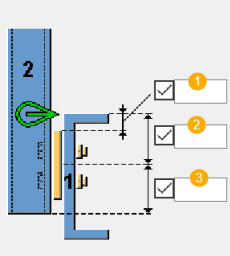


	Description
1	Connection plate

Picture tab

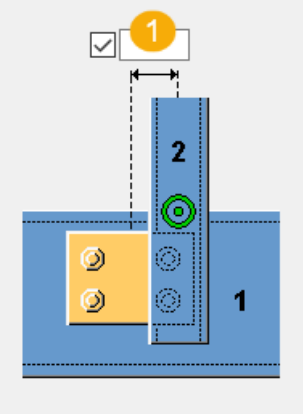
Use the **Picture** tab to control the connection dimensions.

Dimensions



	Description	Default
1	Connection plate distance from the main part upper edge Positive values make the plate smaller.	
2	Vertical position Define the uppermost bolt position as a distance from the stanchion top edge to the uppermost bolt.	76 mm
3	Fit distance Define the stanchion cut level as a distance from the uppermost bolt to the stanchion bottom.	Cut at the bottom level

Plate offset



	Description
1	Horizontal offset of the connection plate

Parts tab

Use the **Parts** tab to define the part properties.

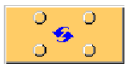
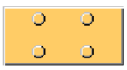
Part

Optionq	Description	Default
Connection plate	Thickness and width of the connection plate.	For rectangular plates: thickness `10 mm` and width `120 mm`.

Parameters tab

Use the **Parameters** tab to define the connection plate orientation and chamfers.

Plate type

Option	Description
	Default Rectangular AutoDefaults can change this option.
	Rectangular
	Diamond

Option	Description
	Triangular
	Circular
	Rectangular by bolts The plate size is determined by the values you enter on the **Bolts** tab.
	Circular by bolts The plate size is determined by the values you enter on the **Bolts** tab.

Plate orientation in skewed situations

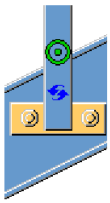
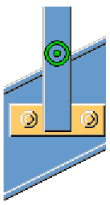
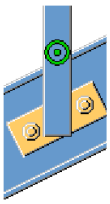
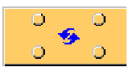
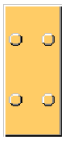
Option	Description
	Default Perpendicular to the secondary part AutoDefaults can change this option.
	Perpendicular to the secondary part
	Oriented to the main part

Plate orientation

Option	Description
	Default Horizontal AutoDefaults can change this option.

Option	Description
	Perpendicular
	Horizontal

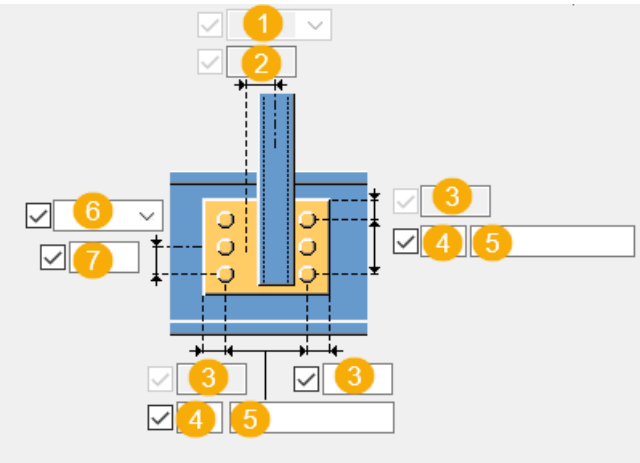
Chamfer type

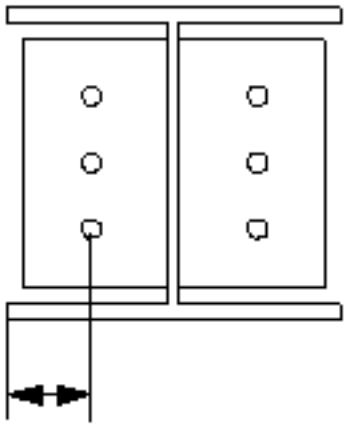
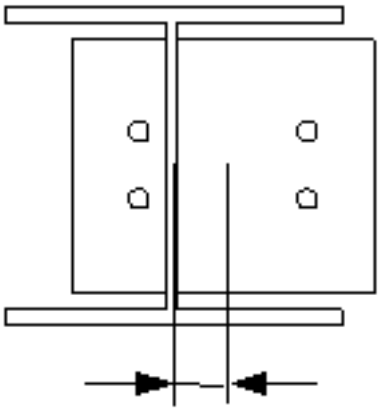
Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer Define the horizontal and vertical chamfer dimensions.
	Round chamfer Define the chamfer radius.

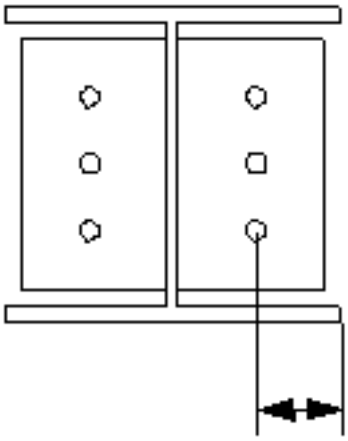
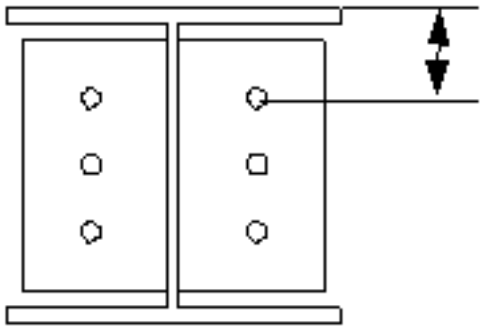
Bolts tab

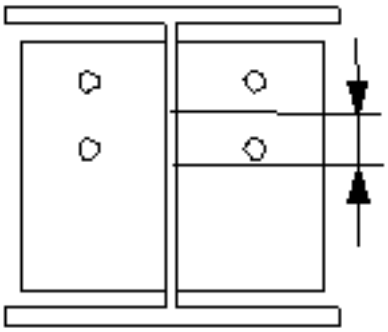
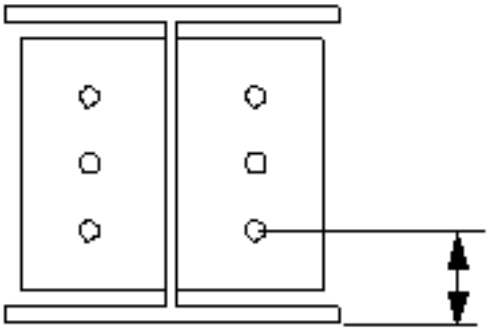
Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt.  • **Middle**: From the center line of the secondary part to the center line of the bolts. 

	Description
	• **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Dimension for vertical bolt group position.

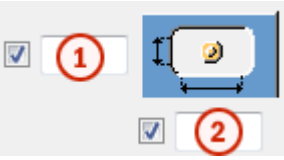
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

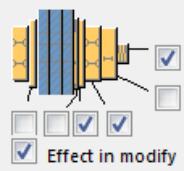


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Multiple beam railing (S84)

Multiple beam railing (S84) creates stanchions and railings connected on one or more beams.

Objects created

- Handrail (top rail)
- Knee rail (bottom rail)
- Stanchions
- Toe plates
- Welds

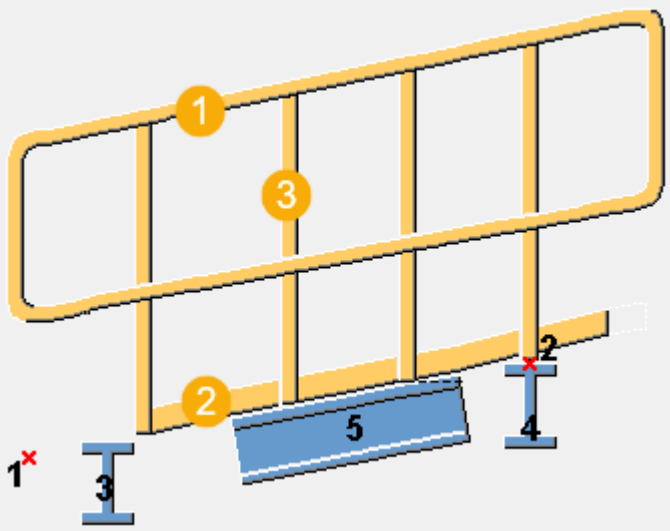
Use for

Situation	Description
	Railing created on multiple beams.

Selection order

1. Pick the first position for the railing.
2. Pick the second position for the railing.
3. Select the first beam.
4. Select the second beam, and the subsequent beams if needed.
5. Click the middle mouse button to create the railing.

Part identification key

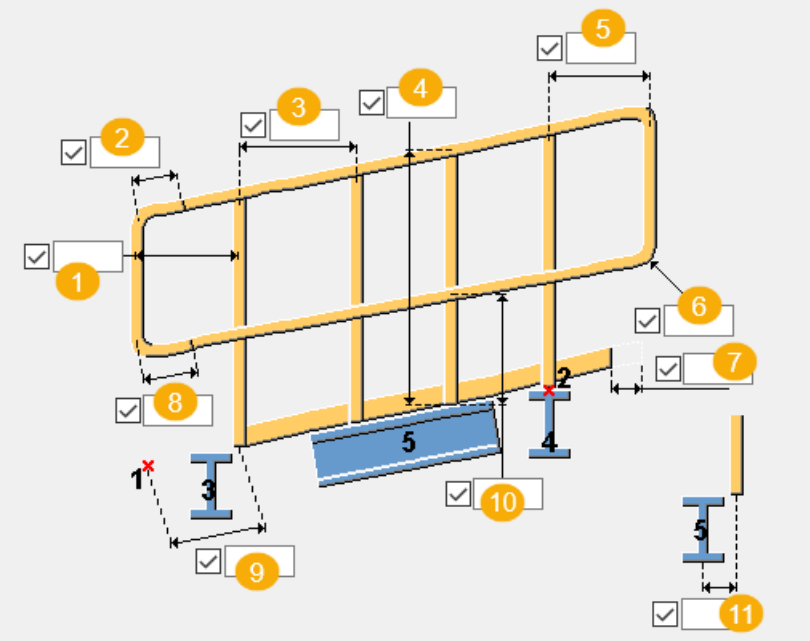


	Description
1	Railing Top rail and bottom rail are created.
2	Toe plate
3	Stanchion

Picture tab

Use the **Picture** tab define the railing dimensions.

Railing dimensions



	Description
1	Full horizontal length of the closure from the first or the last stanchion.
2	Horizontal length of the closure in the top rail and where the bending is created in the closure.
3	Horizontal distance between the stanchions.
4	Distance between the top rail and the beam.
5	Full horizontal length of the closure from the first or the last stanchion.
6	Bending angle
7	Toe plate start/end distance from the start/end of the railing.
8	Horizontal length of the closure in the bottom rail and where the bending is created in the closure.
9	Horizontal distance to the first stanchion from the first input point. This is the position of the first stanchion from the first input point.
10	Distance between the beam and bottom rail.
11	Vertical distance between the beam center and stanchion center.

Parts tab

Use the **Parts** tab control the railing part properties.

Parts

Option	Description
Stanchion profile	Select the stanchion profile from the profile catalog.
Hand rail profile	Select the hand rail profile from the profile catalog.
Knee rail profile	Select the knee rail profile from the profile catalog.
Toe plate	Thickness and height of the toe plate.

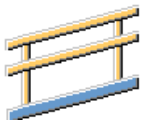
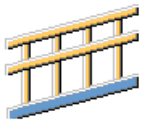
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

Use the **Parameters** tab to define whether stanchions and toe plates are created, and the positions of stanchions, railings, and railing cuts.

Option	Description
Stanchion product name	Enter the stanchion name.
Bend product name	Enter the bend name.
Toe plate max length	Define the maximum length of the toe plate.
Create assembly	Select whether to create an assembly of the parts.

Stanchion creation

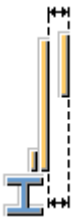
Option	Description
	Stanchions are not created.
	Stanchions are created.

Railing cut positions

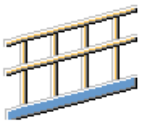
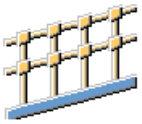
Select the railing cut position and define the cut dimension.

Option	Description
	Railing is cut at the second stanchion.
	Railing is cut at the first stanchion.
	Railing is cut at each stanchion.

Stanchion and railing position

Option	Description
	Stanchions and railings are created on the left side of the beams.
	Stanchions and railings are created on the right side of the beams.

Toe plate creation

Option	Description
	Toe plates are not created.
	Toe plates are created.

Welds

Click the link below to find out more:

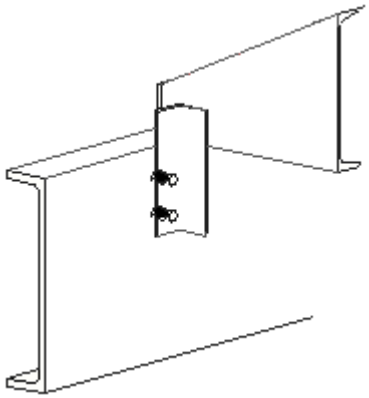
Stringer to channel (127)

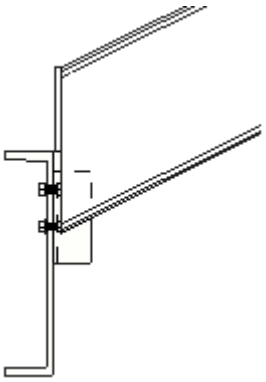
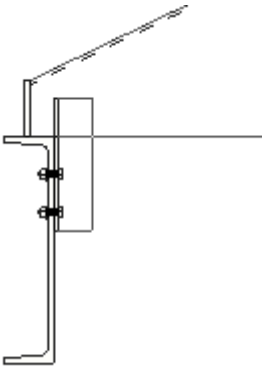
Stringer to channel (127) connects a stringer to a channel with an angle profile. The angle profile is welded to the stringer and bolted to the channel. A vertical plate is welded to the end of the stringer.

Objects created

- Angle plate
- Vertical plate (optional)
- Bolts
- Welds
- Cuts to shape the end of the stringer

Use for

Situation	Description
	The angle profile is bolted to the channel and welded to the stringer. The stringer is cut horizontally to the top level of the channel.

Situation	Description
	No horizontal cut in the stringer.
	A vertical plate is created to the end of the stringer.

Limitations

Stringer to channel (127) works only if the connecting profile is an angle.

Before you start

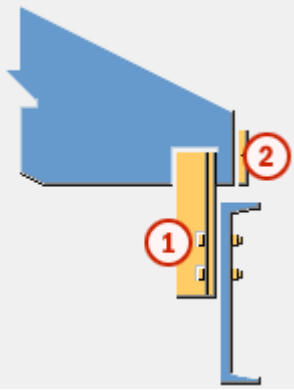
Create a stringer and a channel.

Selection order

1. Select the main part (a channel).
2. Select the secondary part (a stringer).

   The connection is created automatically when the secondary part is selected.

Part identification key

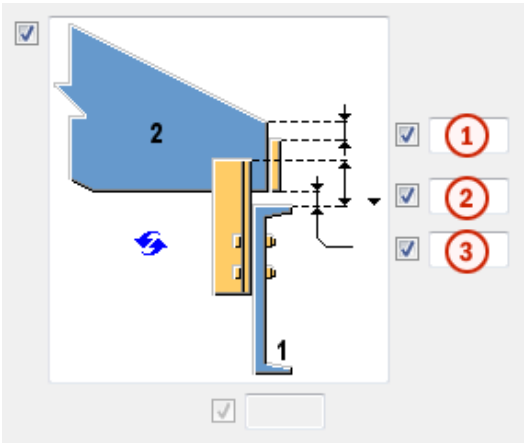


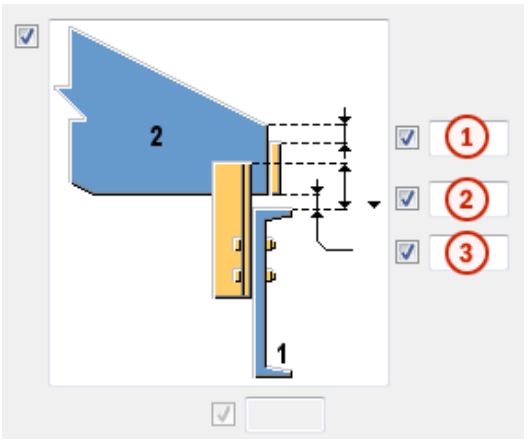
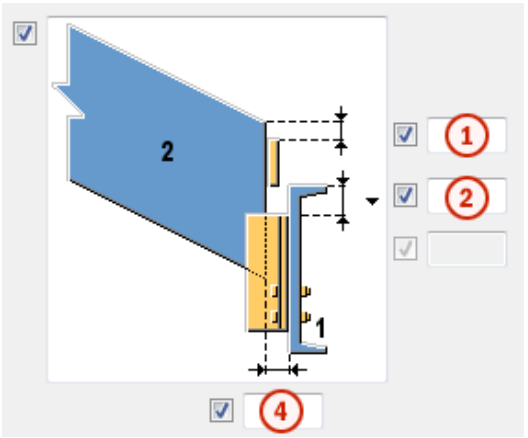
	Part
1	Angle profile
2	Vertical plate

Picture tab

Use the **Picture** tab to define the position of the angle profile and the vertical plate, and whether the stringer is cut.

Stringer cut

Option	Description
	Option 1 Default

Option	Description
	Option 2 The stringer is cut horizontally to the top level of the channel.
	Option 3 No horizontal cut in the stringer.

	Description	Default
1	Position of the vertical plate. Define the vertical distance from the stringer top edge to the vertical plate top.	`0`
2	Position of the angle profile. Define the vertical distance from the channel top to the angle profile top.	Option 1 or 2: • metric: `65 mm` • imperial: `2"1/2` Option 3: • `0`
3	Location of the stringer cut. Define the vertical distance from the channel top edge to the plane where the stringer is cut horizontally and the bottom of the vertical plate is positioned.	• metric: `12 mm` • imperial: `1/2"`

	Description	Default
4	Location of the stringer cut. Define the horizontal distance from the channel web to the stringer.	• metric: `12 mm` • imperial: `1/2"`

Parts tab

Use the **Parts** tab to control the properties of the vertical plate and the angle profile.

Vertical plate and angle profile

Option	Description	Default
Vertical plate	Vertical plate thickness. If you set the value to 0, no vertical plate is created.	metric: `10 mm` imperial: `3/8"` The default name is `PROFILE`.
Angle	Define the angle profile by selecting it from the profile catalog. If you do not select an L profile, the connection will not be created.	metric: `L75*6` imperial: `L3X3X1/4` The default name is `ANGLE`.

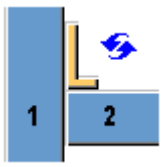
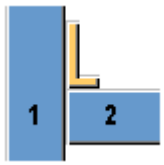
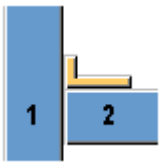
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

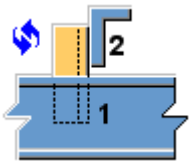
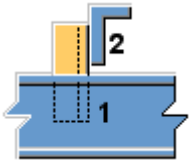
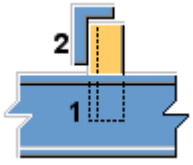
Use the **Parameters** tab to control the leg orientation and the position of the angle profile.

Angle profile leg orientation

If the legs of the angle profile are of uneven length, you can switch their position.

Option	Description
	Default The longer leg of the angle profile is connected to the channel. AutoDefaults can change this option.
	The longer leg of the angle profile is connected to the channel.
	The longer leg of the angle profile is connected to the stringer.

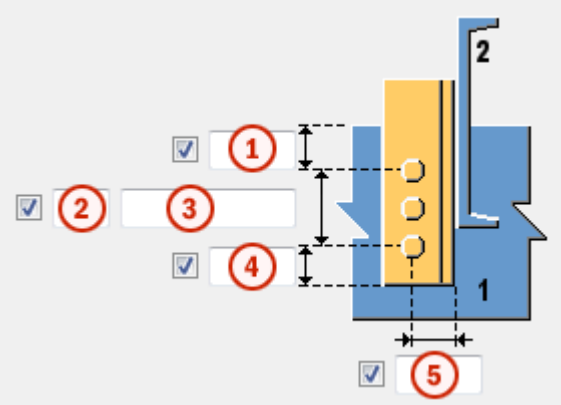
Angle profile position

Option	Description
	Default Angle profile is on the outer surface of the channel. AutoDefaults can change this option.
	Angle profile is on the outer surface of the channel.
	Angle profile is on the inner surface the channel.

Bolts tab

Use the **Bolts** tab to control the bolts that connect the angle profile to the channel.

Bolt group dimensions



	Description	Default
1	Vertical distance from the top of the channel to the center of the top bolt.	metric: `75 mm` imperial: `3"`
2	Number of bolts.	`2`
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.	metric: `75 mm` imperial: `3"`
4	Vertical distance from the bottom of the angle profile to the center of the bottom bolt.	metric: `40 mm` imperial: `1"1/2`
5	Horizontal distance from the stringer edge to the center line of the bolts.	metric: `40 mm` imperial: `1"1/2`

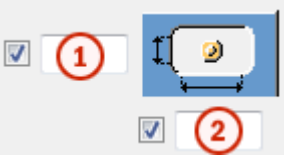
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	

Option	Description	Default
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

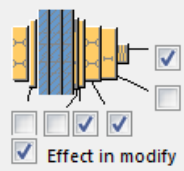


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Stair base detail (1038)

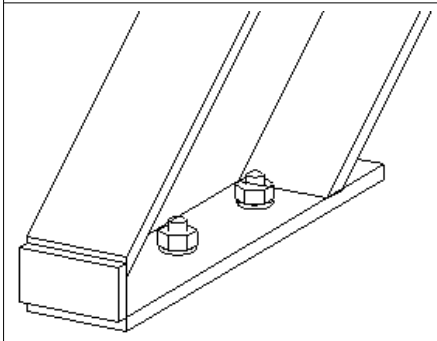
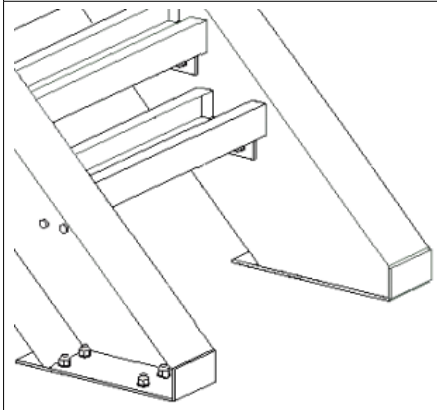
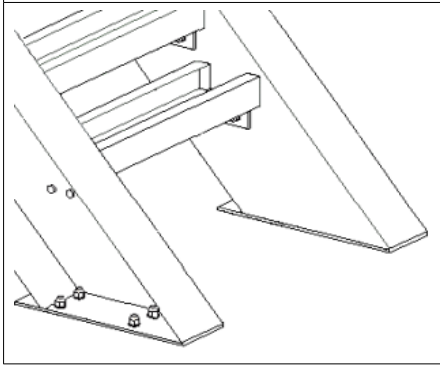
Stair base detail (1038) creates a bolted horizontal plate and an optional vertical plate at a picked point on a stringer. The plates are welded to the stringer.

Objects created

- Horizontal plate
- Vertical plate (optional)
- Bolts
- Welds

- Cuts to shape the end of the stringer

Use for

Situation	Description
	Stair base detail with horizontal and vertical plates.
	Stair base detail with horizontal and vertical plates.
	Stair base detail with horizontal plates.

Limitations

Stair base detail (1038) works only if the **Up direction** is set to **+z** on the **General** tab. It does not work if the **Up direction** is set to **auto**.

Before you start

Create a stringer.

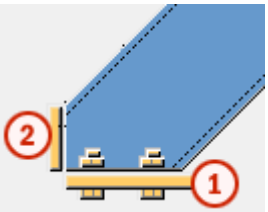
Selection order

1. Select the stringer.

2. Pick a point on the stringer.

   The detail is created automatically.

Part identification key

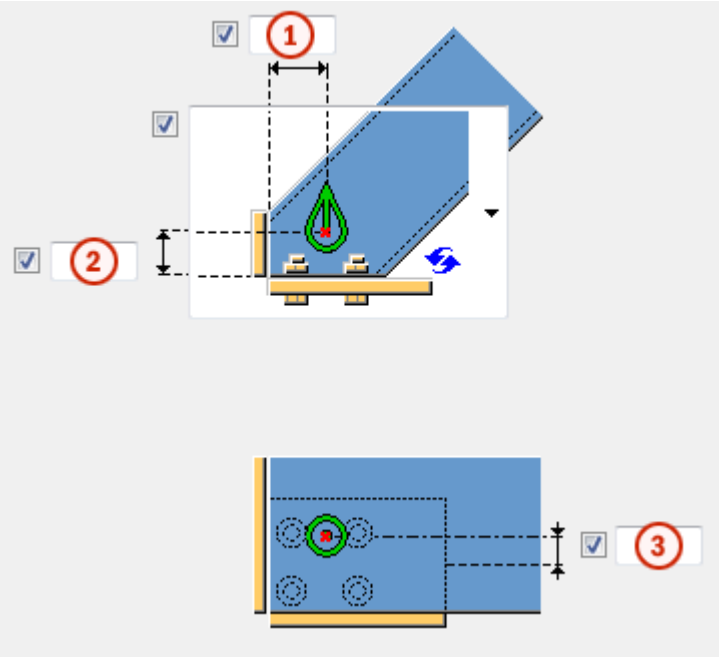


	Part
1	Horizontal plate
2	Vertical plate

Picture tab

Use the **Picture** tab to control the horizontal and vertical plate positions.

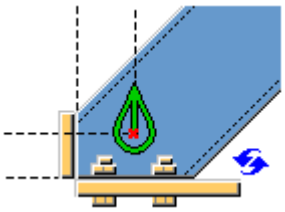
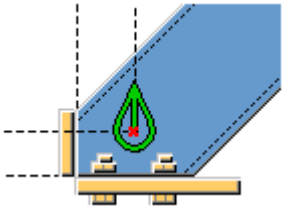
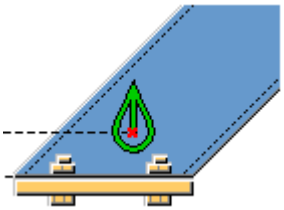
Plate positions



	Description	Default
1	Plate position. Define the horizontal distance from the picked point to the inner face of the vertical plate.	metric: `31 mm` imperial: `1"1/4`

	Description	Default
2	Plate position. Define the vertical distance from the picked point to the top face of the horizontal plate.	0 mm
3	Horizontal plate offset relative to the component's up direction.	0 mm

Vertical plate creation

Option	Description
	Default. Vertical plate is created.
	Vertical plate is created.
	No vertical plate is created.

Parts tab

Use the **Parts** tab to control the horizontal and vertical plate properties.

Horizontal plate/Vertical plate

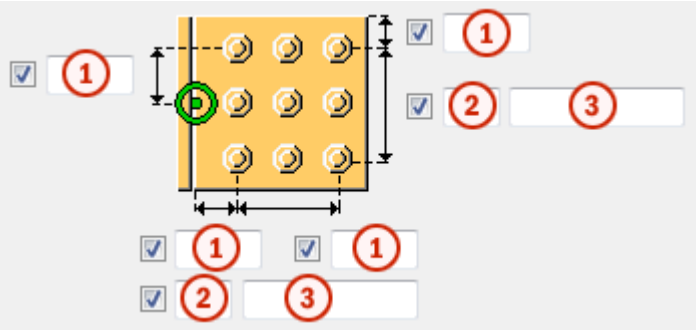
Option	Description
Horizontal plate **Vertical plate**	Plate thickness, width and height. The default name is PLATE.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Bolts tab

Use the **Bolts** tab to control the bolt properties in the horizontal plate. You can select to create bolts or studs. By default, bolts are created.

Bolt group dimensions



	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.

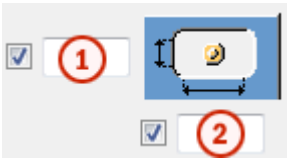
	Description
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



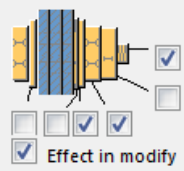
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Bolting direction

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1

Option	Description
	Bolting direction 2

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

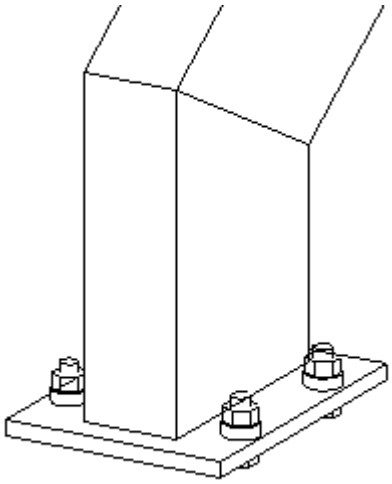
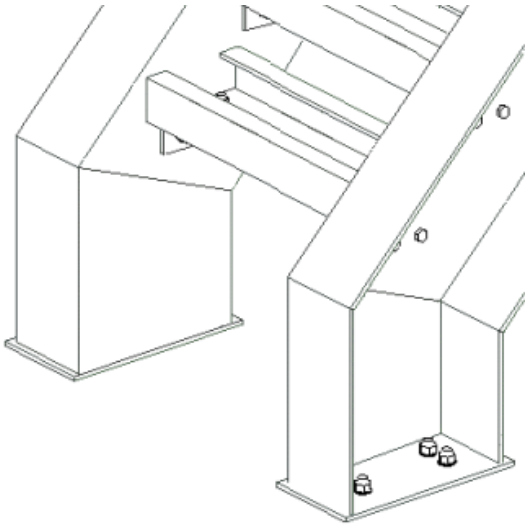
Stair base detail (1039)

Stair base detail (1039) creates a bolted horizontal plate at the picked point on the stringer. The plate is welded to the stringer.

Objects created

- Horizontal plate
- Bolts
- Welds
- Cuts to shape the end of the stringer

Use for

Situation	Description
	Stair base detail with a horizontal plate.
	Stair base detail with a horizontal plate.

Before you start

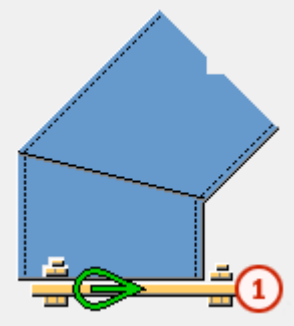
Create a stringer.

Selection order

1. Select the stringer.
2. Pick a point on the stringer.

   The detail is created automatically.

Part identification key

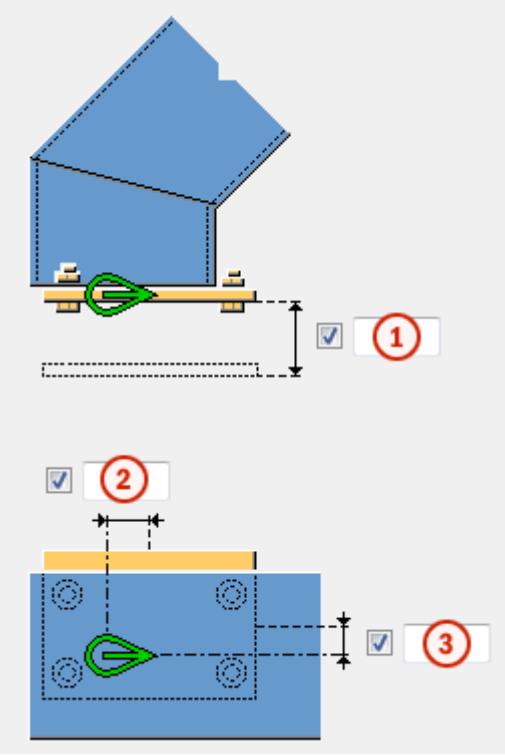


	Part
1	Horizontal plate

Picture tab

Use the **Picture** tab to control the horizontal plate position.

Horizontal plate position



	Description
1	Plate position. Define the vertical distance from the picked point to the top face of the horizontal plate.
2	Horizontal plate offset in the z direction, relative to the component's up-direction.
3	Horizontal plate offset in the y direction, relative to the component's up-direction.

Parts tab

Use the **Parts** tab to control the horizontal plate properties.

Horizontal plate

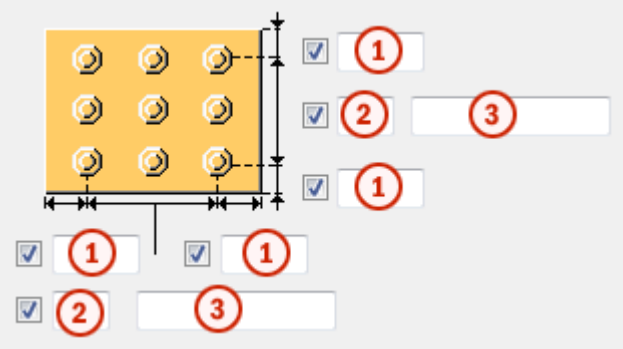
Option	Description
Horizontal plate	Plate thickness, width and height. The default name is `PLATE`.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Bolts tab

Use the **Bolts** tab to control the bolt properties in the horizontal plate. You can select to create bolts or studs. By default, bolts are created.

Bolt group dimensions



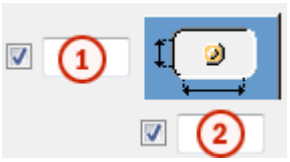
	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

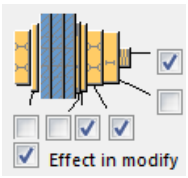


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

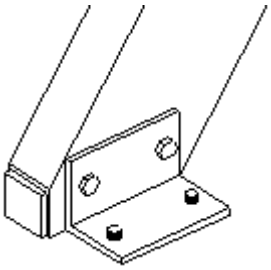
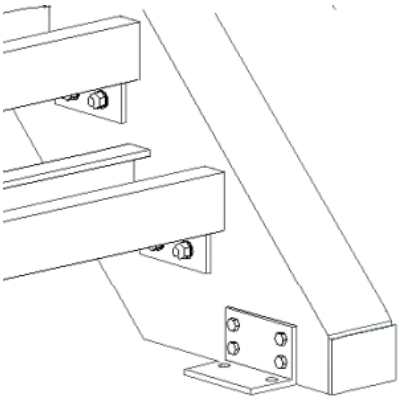
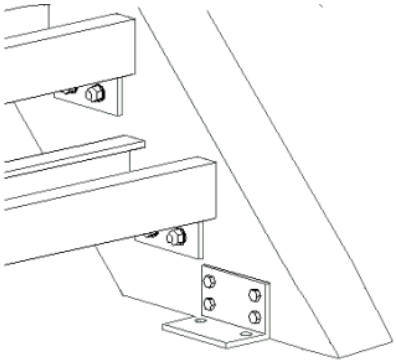
Stair base detail (1043)

Stair base detail (1043) creates a bolted clip angle and an optional vertical plate (nose plate) at a picked point on a stringer. The clip angle is bolted and the vertical plate is welded to the stringer.

Objects created

- Clip angle
- Vertical plate (optional)
- Bolts
- Welds
- Cuts to shape the end of the stringer

Use for

Situation	Description
	Stair base detail with a clip angle and a vertical plate.
	Stair base detail with a clip angle and a vertical plate.
	Stair base detail with a clip angle.

Limitations

Stair base detail (1043) works only if the **Up direction** is set to **+z** on the **General** tab. It does not work if the **Up direction** is set to **auto**.

Before you start

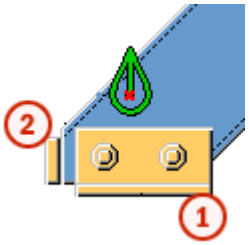
Create a stringer.

Selection order

1. Select the stringer.
2. Pick a point on the stringer.

   The detail is created automatically.

Part identification key

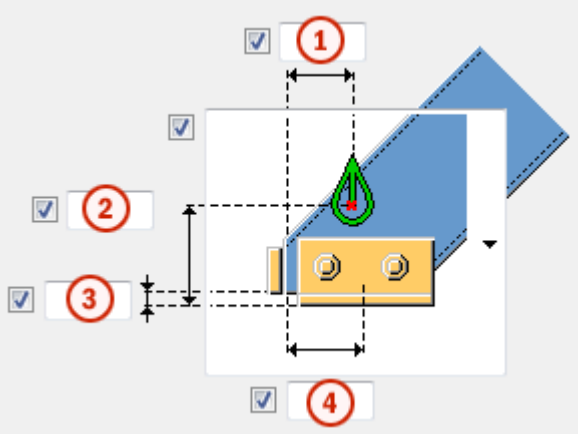


	Part
1	Clip angle
2	Vertical plate

Picture tab

Use the **Picture** tab to control the clip angle and the vertical plate positions.

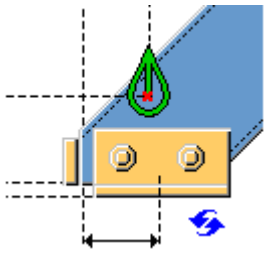
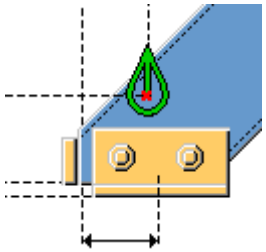
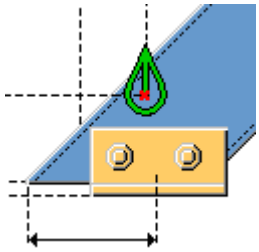
Clip angle position



	Description	Default
1	Plate position. Define the horizontal distance from the picked point to the inner face of the vertical plate.	metric: `31 mm` imperial: `1"1/4`
2	Clip angle vertical position. Define the vertical distance from the picked point to the bottom of the clip angle.	metric: `170 mm` imperial: `6"11/16`
3	Height of the stringer cut.	metric: `12 mm` imperial: `1/2"`

	Description	Default
4	Clip angle horizontal position. Define the horizontal distance from the clip angle center line to the inner face of the vertical plate.	metric: `73 mm` imperial: `2"7/8`

Vertical plate creation

Option	Description
	Default Vertical plate is created. AutoDefaults can change this option.
	Vertical plate is created.
	No vertical plate is created.

Parts tab

Use the **Parts** tab to control the vertical plate and the clip angle properties.

Vertical plate and angle cleat profile

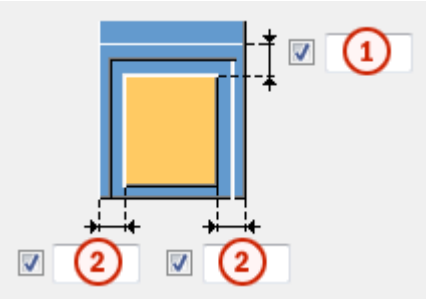
Option	Description
Vertical plate	Plate thickness, width and height. The default name is `PLATE`.
Angle cleat profile	Clip angle profile by selecting it from the profile catalog. The default name is `CLEAT`.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Parameters tab

Use the **Parameters** tab to control the vertical plate and the clip angle positions.

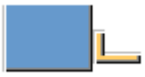
Vertical plate position



	Description
1	Dimension from the top of the vertical plate to the front edge of the stringer.
2	Dimension from the edges of the vertical plate to the edges of the stringer.

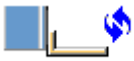
Clip angle location

Select on which side of the stringer the clip angle is created.

Option	Description
	Default Clip angle is created on the left side of the stringer. AutoDefaults can change this option.
	Clip angle is created on the right side of the stringer.
	Clip angle is created on the left side of the stringer.

Clip angle position

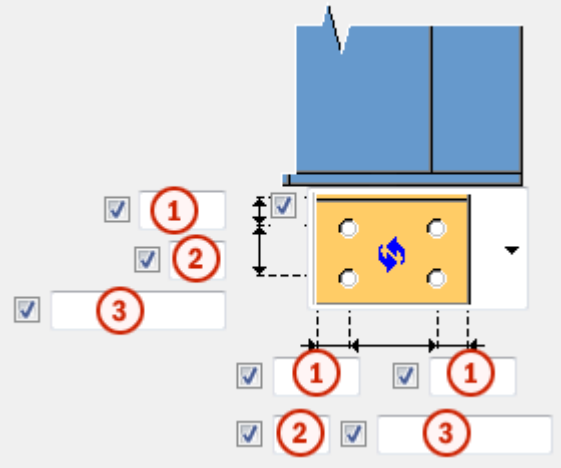
If the legs of the clip angle are of uneven length, you can switch their position.

Option	Description
	Default The shorter leg of the clip angle is connected to the stringer. AutoDefaults can change this option.
	The shorter leg of the clip angle is connected to the stringer.
	The longer leg of the clip angle is connected to the stringer.

PBolts tab

Use the **PBolts** tab to control how the clip angle is attached to the base.

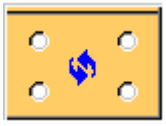
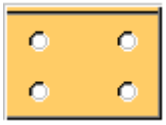
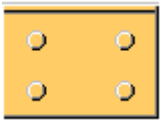
Bolt group dimensions



	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Clip angle attachment

Select how the clip angle is attached to the base.

Option	Description
	Default Holes are created. AutoDefaults can change this option.
	Holes are created.
	Bolts and holes are created.
	No holes or bolts are created.

Create as

Use this option to switch between holes and custom components. Note that you cannot use this option if you have selected that both bolts and holes are created, or that no holes or bolts are created.

Select the custom component from the **Applications & components** catalog and define the custom settings, up direction, rotation, and anchor length.

Bolt basic properties

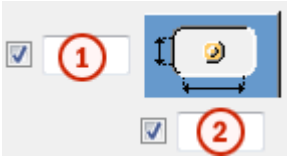
Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Slotted holes

You can define slotted, oversized, or tapped holes.



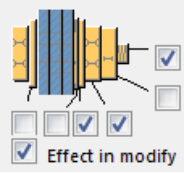
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.

Option	Description	Default
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

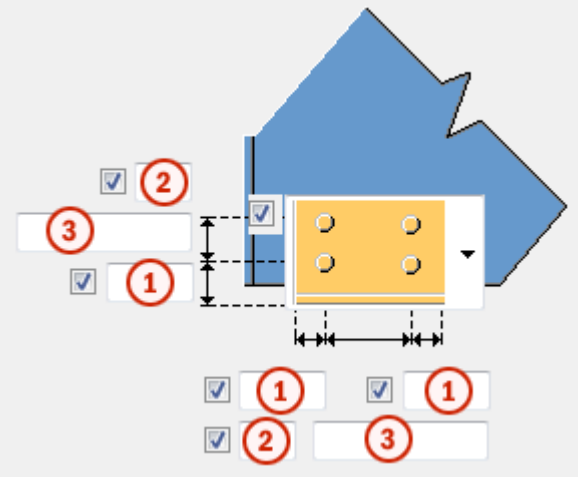
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

SBolts tab

Use the **SBolts** tab to control how the clip angle is attached to the stringer.

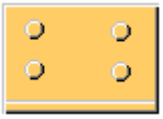
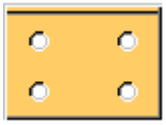
Bolt group dimensions



	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Clip angle attachment

Select how the clip angle is attached to the stringer.

Option	Description
	Default No bolts are created. AutoDefaults can change this option.
	No bolts are created.
	Bolts are created.
	Holes are created.

Bolt basic properties

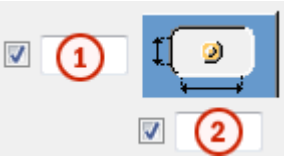
Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Slotted holes

You can define slotted, oversized, or tapped holes.



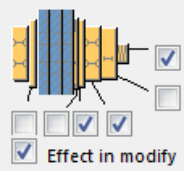
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Create as

Use this option to switch between holes and custom components. Note that you cannot use this option if you have selected that both bolts and holes are created, or that no holes or bolts are created.

Select the custom component from the **Applications & components** catalog and define the custom settings, up direction, rotation, and anchor length.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

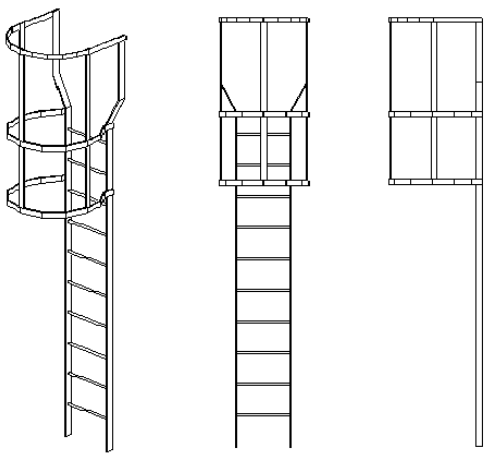
Ladder (S35)

Ladder (S35) creates a vertical step-through ladder, with an option for a safety cage.

Objects created

- Stringers
- Hoops
- Rungs
- Vertical bars

Use for

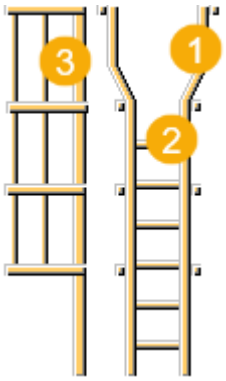
Situation	Description
	Ladder with safety cage and vertical step.

Selection order

1. Pick a point to indicate the top point of the ladder.
2. Pick a point to indicate the elevation of the ladder.

   The ladder is created automatically when you pick the second point.

Part identification key

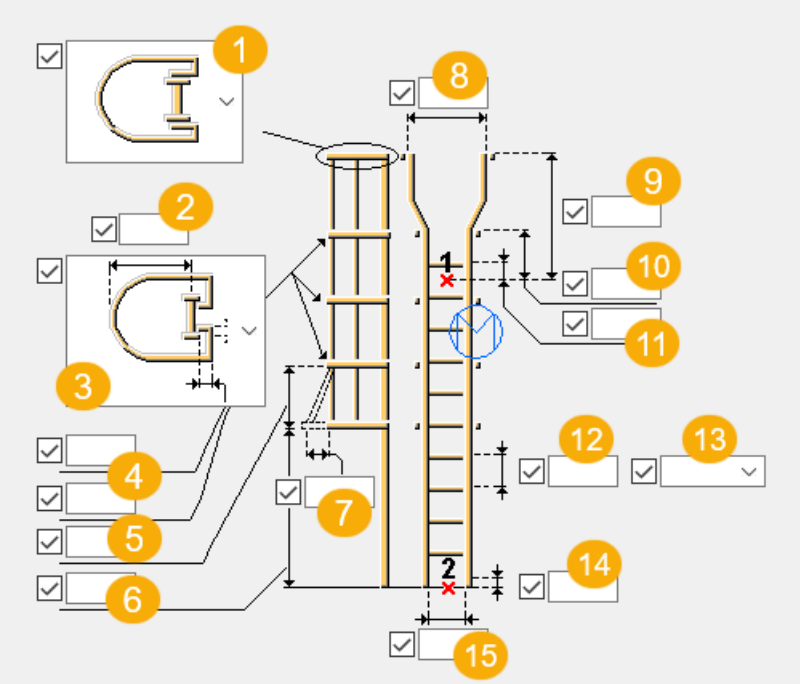


	Description
1	Stringer
2	Rung
3	Cage

Picture tab

Use the **Picture** tab to define the ladder geometry and dimensions.

Ladder dimensions



	Description	Default
1	Direction from which the ladder is accessed. The options are: left, right, or normal (default).	
2	Hoop circle offset from the stringer center line.	760 mm
3	Shape of the cage and how it connects to the ladder. The options are: U type 1 (default), U type 2, and O type.	
4	Cage connection options The first option is used only for U type 1. Both options are used for U type 2.	First option: 75 Second option: 40
5	Maximum spacing of hoops. The hoops are evenly spaced below the stringer flare. Only one hoop is placed on the flared stringer.	900 mm
6	Position (z-coordinate) of the lowest hoop bottom edge. If the value is greater than the floor to floor dimension, only the top hoop is	2500 mm

	Description	Default
	placed with no bars. If the value is greater than the floor to floor dimension plus stringer projection, then no hoops are placed.	
7	Lowest hoop extra diameter dimension.	
8	Inside diameter of the hoops. Stringer flare is taken from the diameter of the hoops.	750 mm
9	Stringer projection Height from the upper top of steel to the flared stringer top edge.	1100
10	Kicker height above the upper top of steel at which the ladder stringer flare.	200
11	Thickness of the top floor above the top of steel. The top rung is placed level with the top of flooring thickness.	0
12	Maximum spacing of rungs. Rungs are evenly spaced between the top and bottom floors.	300 mm
13	Distance between the rungs. Select either equal or exact distance.	**Equal**
14	Thickness of the bottom floor above the lower top of steel. Bottom rung spacing is taken from the lower top of steel plus bottom floor.	0
15	Width of the ladder.	450 mm

Parts tab

Use the **Parts** tab to define the part properties.

Parts

Option	Description	Default
Stringers	Select the stringer profile from the profile catalog.	The default value is flat 10*65 ("PLT10*65").
Rungs	Select the rung profile from the profile catalog.	The default value is round bar 20 ("D20").

Option	Description	Default
Top Hoops	Select the top hoop profile from the profile catalog.	The default value is flat `10*50 ("PLT10*50").`
Middle Hoops	Select the middle hoop profile from the profile catalog.	The default value is flat `10*50 ("PLT10*50").`
Bottom Hoop	Select the bottom hoop profile from the profile catalog.	The default value is flat `10*50 ("PLT10*50").`
Vertical Bars	Select the vertical bar profile from the profile catalog.	The default value is flat `10*50 ("PLT10*50").`

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

Use the **Parameters** tab to define the ladder properties and position.

Ladder position

Option	Description	Default
Vertical position	Vertical position (page 624) of the ladder	**Middle**
Vertical offset	Vertical offset of the ladder	`0.0`
Horizontal position	Horizontal position (page 626)of the ladder	**Middle**

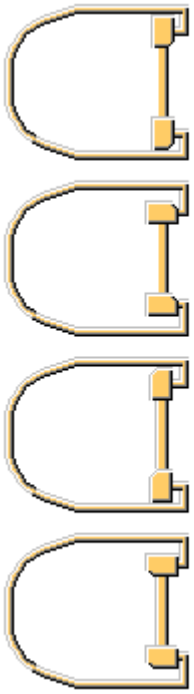
Option	Description	Default
Horizontal offset	Horizontal offset of the ladder	`0.0`
Class	Class of the ladder	`1`
Create assembly	Select whether to create an assembly or not.	Assembly is created.

Ladder rotation

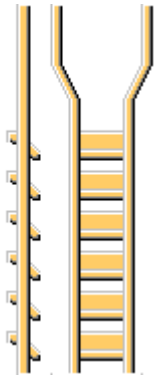
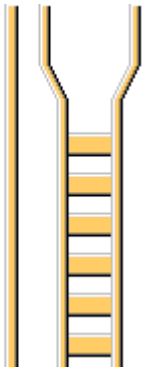
Define how the ladder is rotated. The default value is front.

Stringer rotation

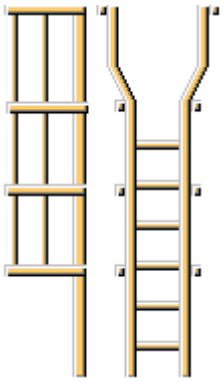
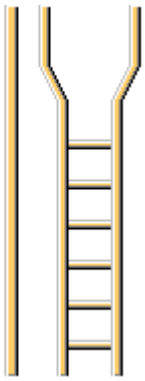
Define how the stringers are rotated. The default value is top.



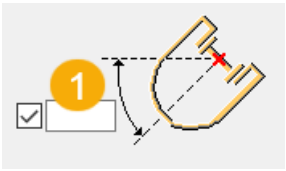
Rung rotation

Option	Description
	Rungs are rotated by 45 degrees. This is the default option.
	Rungs are not rotated.

Cage creation

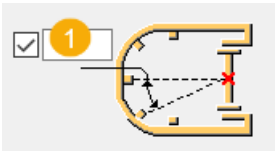
Option	Description
	Cage is created. This is the default option.
	Cage is not created.

Hoop angle



1	Define the hoop angle.

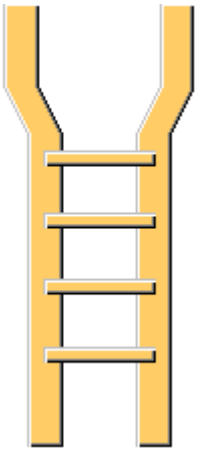
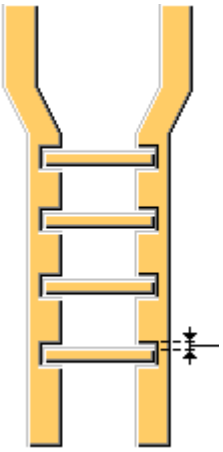
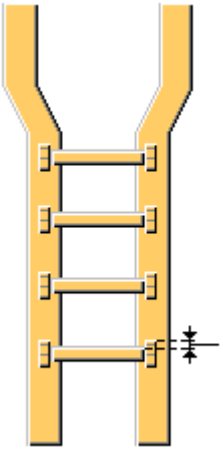
Vertical bar spacing angle

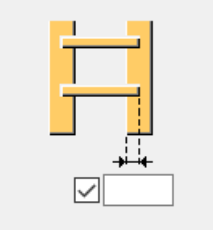


	Description	Default
1	Define the angle between the vertical bars in the round section of the hoop.	30 degrees

Rung stringer part cut

Select whether stringers are cut to avoid clashing.

Option	Description
	Stringers are not cut. This is the default option.
	Stringers are cut. Define the clearance value for the cut.
	Stringers are cut. Rungs are bolted to the stringers. Define the clearance value for the cut.

Option	Description
	Define the depth of the cut into the stringer.

Rung offset

	Description
1	Define the horizontal offset of rungs from the stringer center line.

Cage ladder (S60)

Cage ladder (S60) creates a vertical step-through ladder with an option to create a safety cage.

Objects created

- Rail bars
- Rungs
- Backer channel
- Backer plates
- Supports
- Hoops
- Vertical bars
- Welds

Selection order

1. Pick a point to indicate the top point of the ladder.

2. Pick a point to indicate the elevation of the ladder.

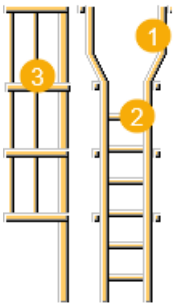
   The second point can be anywhere on the top of the bottom platform. The second point does not need to be directly below the first point but the two points should define a line that is nearly parallel to the current Z axis.

   The ladder is created automatically when you pick the second point.

If the line between the two points is not nearly parallel to the Z axis, the ladder is not created. If the two points are suitable for creating the ladder, the second point is adjusted, if needed, to define a line that is perfectly parallel to the Z axis. The ladder is positioned relative to this reference line.

Before creating the component, we recommend that you set the user coordinate system so that the X axis and Y axis lie in a horizontal plane with the Z axis pointing straight up. The component works with other orientations as well, but regardless of the orientation, the ladder is always positioned exactly parallel to the Z axis.

Part identification key

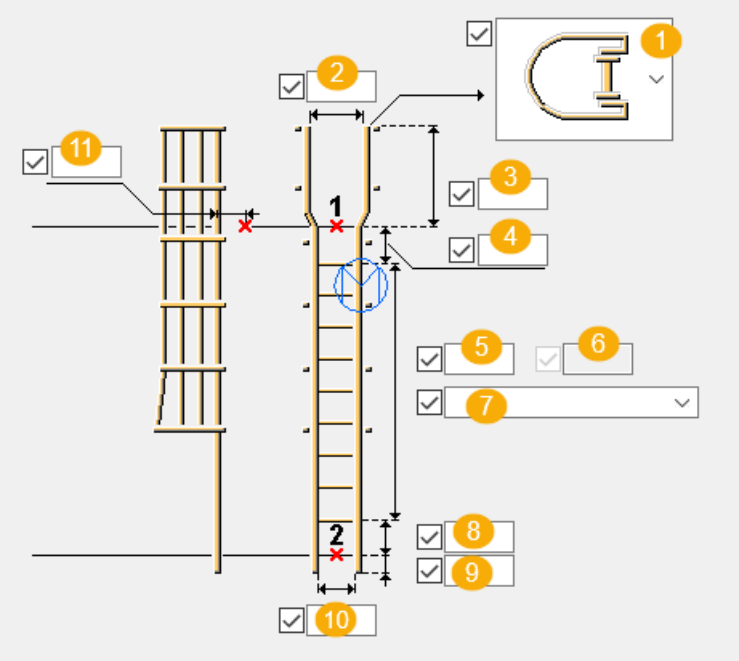


	Description
1	Rail bar
2	Rung
3	Cage

Picture tab

Use the **Picture** tab to define the ladder geometry and dimensions.

Dimensions



	Description
1	Step type Direction from which the ladder is accessed: left side, right side, or step through (default).
2	Inside width of the ladder from the inner edge of one rail to the other The flared width is taken from the inner diameter of the hoops. Flared width is always set to more than the ladder width (**10**). If you set the flared width to less than or equal to the ladder width, the flared width is automatically reset to the ladder width plus 4 inches. If side steps are used, the ladder width is used as the width.
3	Ladder top offset from the upper top of steel to the flared rail bar top edge
4	Top rung offset Defines the distance from the first input point to the center line of the topmost ladder rung.

	Description
5	Number of rungs To define the number of rungs, select either **Default** or **Number of rungs** from the list below the box (**7**). If side steps are used, four additional rungs are automatically added above the top platform at a spacing of 12 inches.
6	Distance between the rungs
7	Select to set either the maximum distance between the rungs, or the exact spacing from the bottom or top rung.
8	Bottom rung offset Defines the distance from the second input point to the center line of the lowest ladder rung.
9	Ladder bottom offset Defines the distance from the bottom platform to the bottom of the ladder rails. A positive value places the bottom of the ladder below the bottom platform.
10	Ladder width Defines the inside width of the ladder from the inner edge of one rail to the other. If you select the step through option (**1**), the portion of the ladder above the top platform will be flared. If you do not define the ladder width, the default value in the applicable `jobdefs.lis` file is used.
11	Toe offset Defines the distance from the center line of the ladder rail to the toe of the top platform. If you do not define this offset, the default value in the applicable `jobdefs.lis` file is used.

Parts tab

Use the **Parts** tab to define the part profiles and properties.

Parts

Option	Description
Rail bars	The bar profile used to form the ladder rails Select the profile from the profile catalog.

Option	Description
Rungs	The rod profile used to form the ladder rungs Select the profile from the profile catalog.
Backer channel	The channel profile used for the backer support channel Select the profile from the profile catalog.
Backer plates	The plate profile used for the bent plates that attach the ladder rails to the backer support channel Select the profile from the profile catalog.
Ladder supports	Select the profile from the profile catalog.
Backside supports	Select the profile from the profile catalog.
Top hoops **Bottom hoop** **Middle hoops**	The bar profile used to form the safety cage hoops Select the profile from the profile catalog.
Vertical bars	The bar profile used to form the vertical cage bars Select the profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Option	Description	Default
Class	Part class number.	

Parameters tab

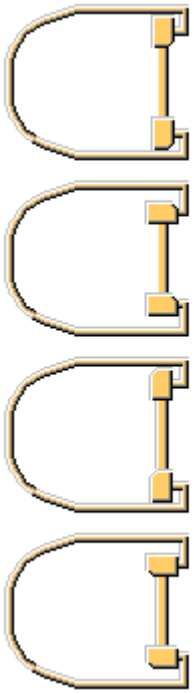
Use the **Parameters** tab to define the ladder properties and position.

Horizontal position and offset

Picture above	Description	Default
Horizontal position	Position of the ladder in the plane of the ladder • **Middle** : the center line of the ladder coincides with the reference line. • **Left** and **Right**: the outside face of the corresponding rail coincides with the reference line.	**Middle**
Horizontal offset	Lateral offset of the ladder in the plane of the ladder For example, an offset of 6" coupled with **Horizontal position** set to **Middle** places the center line of the ladder 6" to the right of the reference line. An offset of -8" with **Horizontal position** set to **Right**, for example, places the outside face of the right-hand rail 8" to the left of the reference line.	0

Stringer rotation

Define how the stringers are rotated. The default value is top.

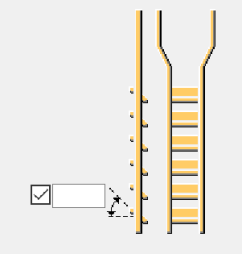


Ladder rotation

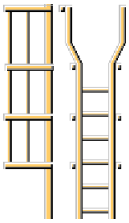
Define how the ladder is rotated. The default value is front.

Rung rotation

Define the rung rotation.

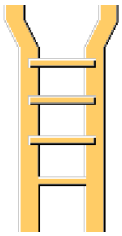


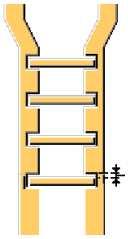
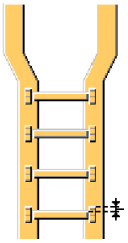
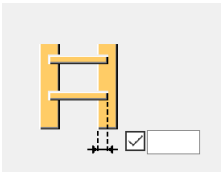
Cage creation

	Default Cage is not created. AutoDefaults can change this option.
	Cage is not created.
	Cage is created.

Rung stringer part cut

Select whether stringers are cut to avoid clashing.

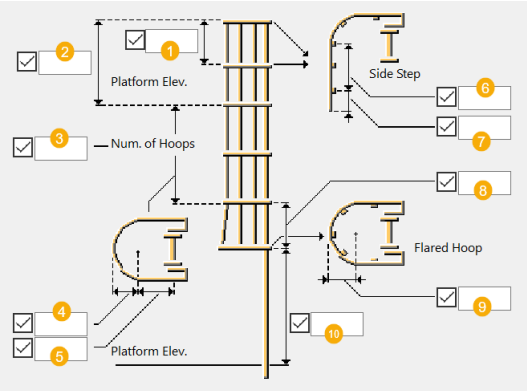
Option	Description
	Stringers are not cut.
	Stringers are not cut.

Option	Description
	Stringers are cut. Define the clearance value for the cut.
	Stringers are cut. Rungs are bolted to the stringers. Define the clearance value for the cut.
	Define the depth of the cut into the stringer.

Cage tab

Use the **Cage** tab to define the size and position of the ladder's safety cage.

Cage dimensions



	Description
1	Distance from the center of the top flared hoop to the next higher hoop in the safety cage
2	Distance from the top platform to the center of the top safety cage hoop

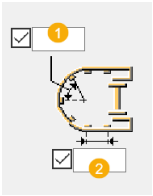
	Description
3	Number of hoops (in addition to the required bottom flared hoop). The default number of hoops is 2. When you have selected that the ladder is accessed from the side, three additional hoops above the top platform are added at a spacing of 1'-10". When you have selected that the ladder has step-through access, two additional hoops are added at 12 inches and 3'-6" above the platform.
4	Hoop radius Defines the outside radius of each safety cage hoop above the flared hoop at the bottom of the cage. The hoop radius at the bottom of the cage is determined by the flared radius. If you do not define the radius, the applicable `jobdefs.lis` file determines the default value. You can define a combination of dimensions, including the hoop radius and ladder width, that will make it impossible to bend and attach the hoops to the outside face of each rail. This applies especially to the top of a step-through ladder where the ladder rails are flared out for improved accessiblity.
5	Cage depth Defines the distance from the center line of the ladder rails to the curvature point for all safety cage hoops. If you do not define the depth, the applicable `jobdefs.lis` file determines the default value.
6	Side step dimension Defines the distance from the ladder's center line to the center line of the outermost vertical cage bar in the side step portion of the cage. When the step type is step through, this dimension is not used.
7	Side step dimension Defines the distance from the center line of the outermost vertical cage bar to the end of the modified safety cage hoops. When the step type is step through, this dimension is not used.
8	Flared hoop space Defines the distance from the center of the bottom flared hoop to the next higher hoop in the safety cage.

	Description
9	Flared radius Defines the outside radius of the flared safety cage hoop at the bottom of the cage. The radius is equal to the hoop radius plus 1.75 inches. The radius of all other hoops is determined in the hoop radius (**4**).
10	Flared hoop offset Defines the distance from the bottom platform to the center of the bottom flared safety cage hoop. If you do not define the offset, the applicable `jobdefs.lis` file determines the default value.

Hoop shape

Option	Description
	U type hoop
	U type hoop
	O type hoop

Vertical bar spacing



	Description
1	Define the angle between the vertical bars in the round section of the hoop.
2	Define the distance between the vertical bars in the straight section of the hoop.

Bolts tab

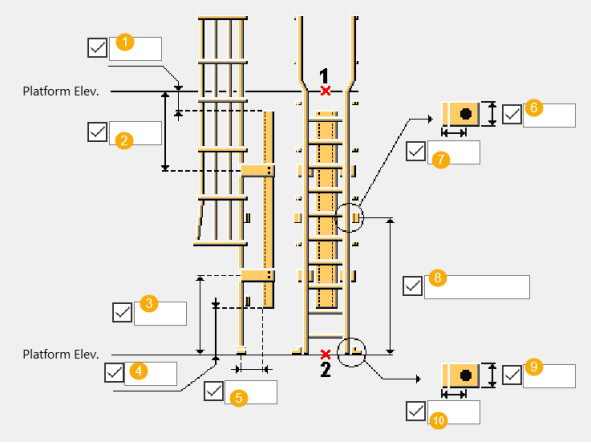
Use the **Bolts** tab to define the bolt properties of the bolts used for backer, ladder, and backside support plates. If a backer channel is not created, the bolt settings are not used.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Supports tab

Use the **Supports** tab to define the backer channel settings.



	Description
1	Top of the channel Top platform elevation distance from the top of the backer channel

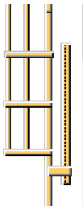
	Description
2	Top support offset Top platform elevation distance to the center line of the upper bent plate used to attach the ladder to the backer channel. If one backer channel support is created, the bottom support offset is used to control the position of the single bent plate.
3	Bottom support offset Bottom platform elevation distance to the center line of the lowest bent plate used to attach the ladder to the backer support channel. If one backer channel support is created, this setting is used to control the position of the single bent plate.
4	Bottom of channel Bottom platform elevation distance from the bottom of the backer channel.
5	Backer offset Distance from the center line of the ladder rail to the nearest face of the backer channel.
6	Vertical top backer support dimension
7	Horizontal top backer support dimension
8	Bottom offset to the middle backer support
9	Vertical bottom backer support dimension
10	Horizontal bottom backer support dimension

Backer channel

Option	Description
	Default Backer channel is not created. AutoDefaults can change this option.
	Backer channel is not created.

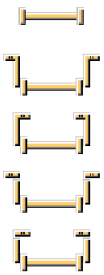
Option	Description
	Backer channel is created to stiffen and support the ladder.

Backer channel supports

Option	Description
	Default Two backer channel supports are created. The bent plate backer supports attach the ladder rails to the backer bar support channel. AutoDefaults can change this option.
	One backer support is created.
	Two backer supports are created.

Top backer support position

Select how the top backer supports are positioned.



Bottom backer supports

Option	Description
	Bottom backer supports are not created.
	Bottom backer supports are created at the outer edges. You can define the vertical and horizontal offset.
	Bottom backer supports are created at the inner edges. You can define the vertical and horizontal offset.

Welds

Click the link below to find out more:

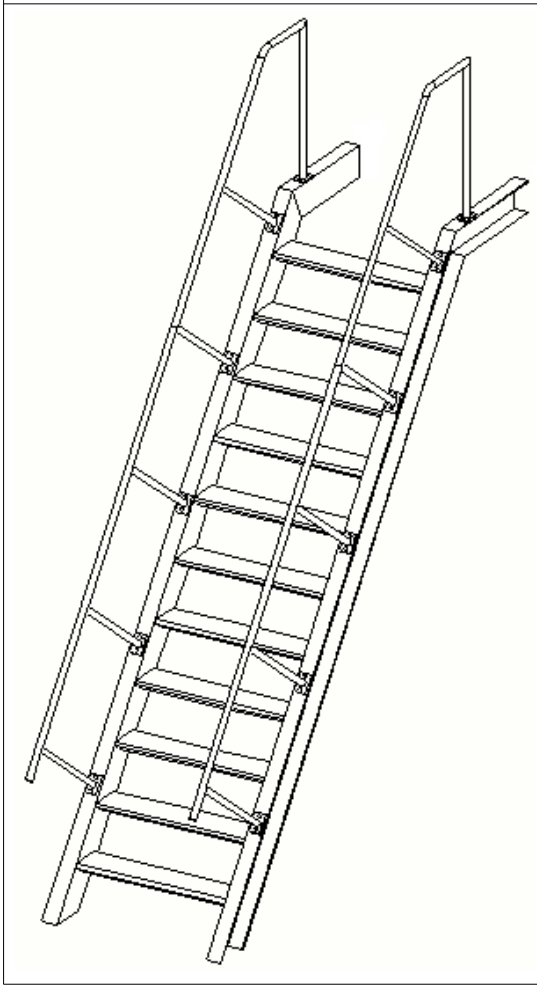
Ship Ladder

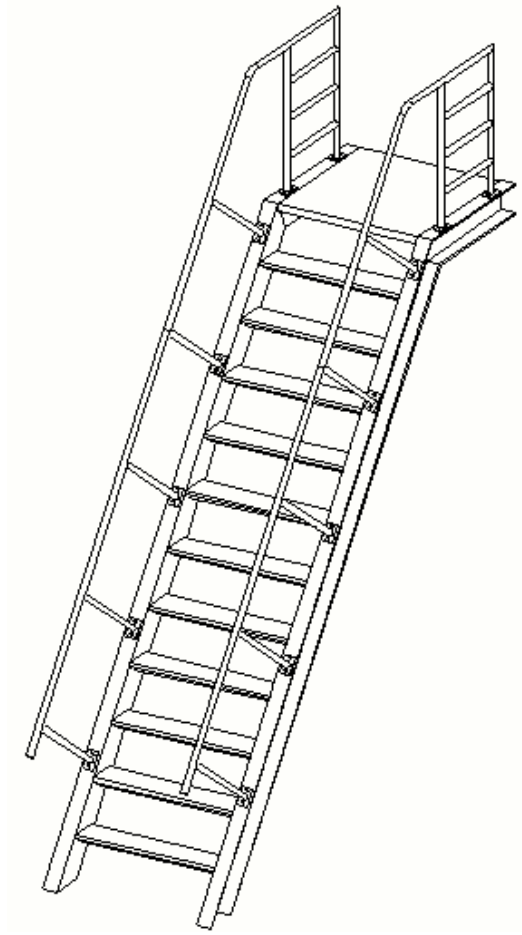
Ship Ladder creates stairs for ships and oil platforms.

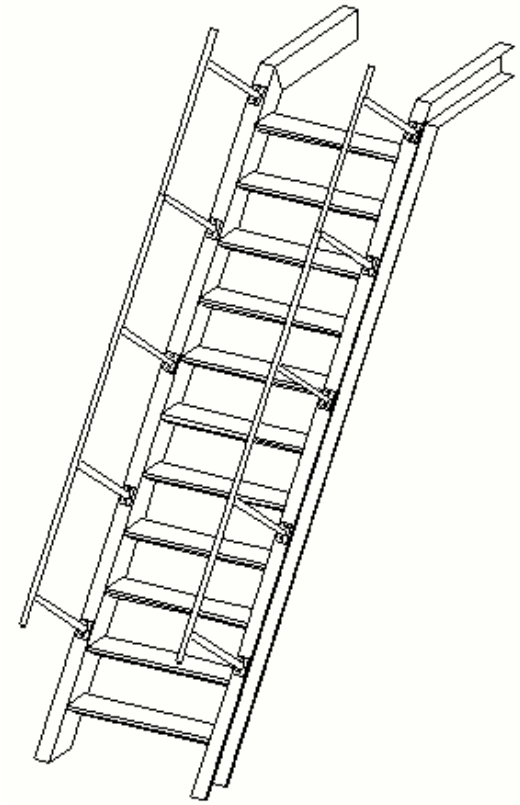
Objects created

- Stanchions
- Stringers
- Steps
- Handrails and elbows
- Middle rails
- Platform
- Platform support
- Bolts
- Welds
- Additional component (optional)

Use for

Situation	Description
	Type 1 Ship ladder with stringers, steps, stanchions, and handrails.

Situation	Description
	Type 2 Ship ladder with stringers, steps, stanchions, platform, and handrails with middle rails.

Situation	Description
	Type 3 Ship ladder with stringers, steps, stanchions, platform, handrails with middle rails.
	Type 4 Ship ladder with stringers, steps, stanchions, and handrails.

Limitations

Ship Ladder uses catalog step profiles to define the step type on the **Parts** tab.

To have an updated step profile list on the **Parts** tab, you need to run the `Steps.exe` program when you use the **Ship Ladder** modeling tool for the first time in your environment, or when you change your Tekla Structures environment.

WARNING With standard settings Tekla Structures overwrites the step profile settings and replaces them with the default values every time when Tekla Structures is started. To prevent Tekla Structures loosing the step profile settings, set `XS_DO_NOT_OVERWRITE_PLUGIN_INP_FILE = TRUE` in `teklastructures.ini` file.

If you are using catalog step profiles and have set `XS_DO_NOT_OVERWRITE_PLUGIN_INP_FILE = TRUE` and you update Tekla Structures, do the following:

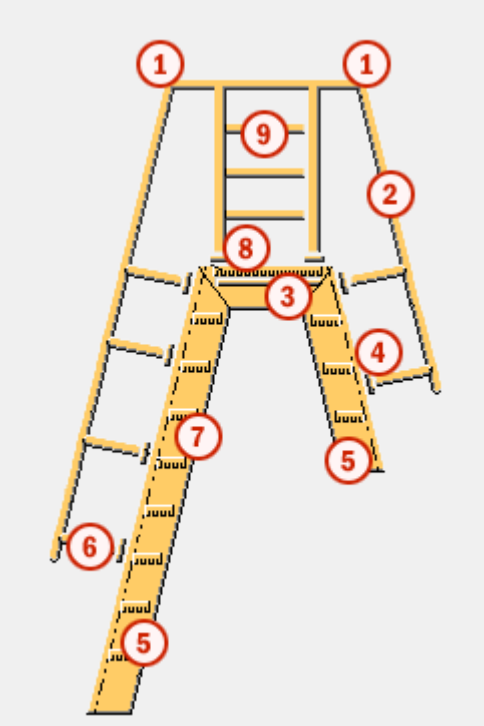
1. Set `XS_DO_NOT_OVERWRITE_PLUGIN_INP_FILE = FALSE` in `teklastructures.ini` file.
2. Update Tekla Structures.
3. Start Tekla Structures.
4. Set `XS_DO_NOT_OVERWRITE_PLUGIN_INP_FILE = TRUE` in `teklastructures.ini` file.
5. Run `Steps.exe`.
6. Restart Tekla Structures

Selection order

1. Pick a point to indicate the bottom level of the stairs.
2. Pick a point to indicate the top level of the stairs.
3. Click the middle mouse button to create the component.

   Points that indicate the bottom level and top level are usually nosing line start/end points.

Part identification key

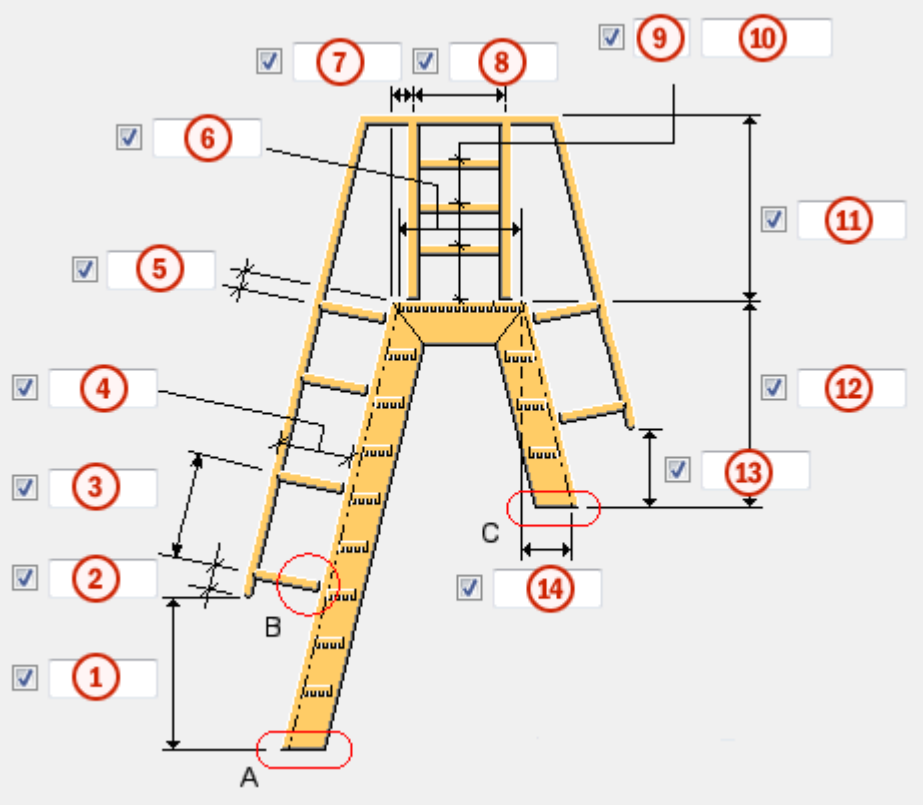


	Part
1	Elbow
2	Handrail
3	Platform support
4	Plate
5	Stringer
6	Stanchion
7	Step
8	Platform
9	Middle rail

Picture tab

Use the **Picture** tab to control the type of the stairs, the dimensions of the parts created, horizontal offset, and horizontal distance between stringers.

Dimensions



	Description	Default
1	Vertical distance between up stringer bottom and handrail bottom.	1000 mm
2	First stanchion offset along handrail, measured from the handrail bottom.	200 mm
3	Maximum distance between middle stanchions.	1000 mm
4	Distance between stringer and handrail.	800 mm
5	Last stanchion offset along handrail, measured from the stringer top.	200 mm
6	• For **Type 1**, **2** and **4**: Define the distance between up nosing line top point and horizontal stringer end. • For **Type 3**: Define the distance between up nosing line top points.	1000 mm

	Description	Default
7	Horizontal offset for the first stanchion, measured from horizontal stringer start. This option is not active for **Type 4**.	`200 mm`
8	Distance between the first and the last horizontal stanchions. This option is not active for **Type 1** or **Type 4**.	`600 mm`
9	Number of middle rails. This option is not active for **Type 1** or **Type 4**.	`3`
10	Middle rail spacing. Use a space to separate middle rail spacing values. Enter a value for each space between middle rails. For example, if there are 3 middle rails, enter 2 values. This option is not active for **Type 1** or **Type 4**.	`Value of option` **3** `divided by number of spaces.`
11	Vertical distance between handrail top and platform top. This option is not active if for **Type 4**.	`1000 mm`
12	Vertical distance between platform top and down stringer bottom. This option is active only for **Type 3**.	`2000 mm`
13	Vertical distance between handrail bottom and down stringer bottom. This option is active only for **Type 3**.	`1000 mm`
14	Horizontal distance between up nosing line end and down nosing line end. This option is active only for **Type 3**.	`The up stringer and down stringers have the same inclination.`

Step and assembly creation

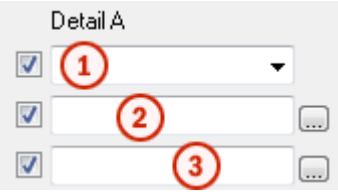
Option	Description
Create top step	Define whether the top step is created.
Create assembly	Define which parts form an assembly. The options are: • **No** No assembly is created.

Option	Description

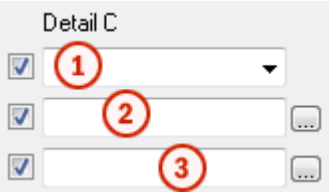
	• **All**
	All parts are included in an assembly.
	• **Stingers/Rails**
	Parts form several assemblies:
	  • Each of the stringer assemblies includes up, horizontal and down stringers.
	  • Each of the rail assemblies includes up, horizontal and down handrails, middle rails, stanchions and plates.
	  • The platform, each platform support and each step form its own assembly.

Detail A and Detail C

Use **Detail A** to connect the up stringer end to an end detail.

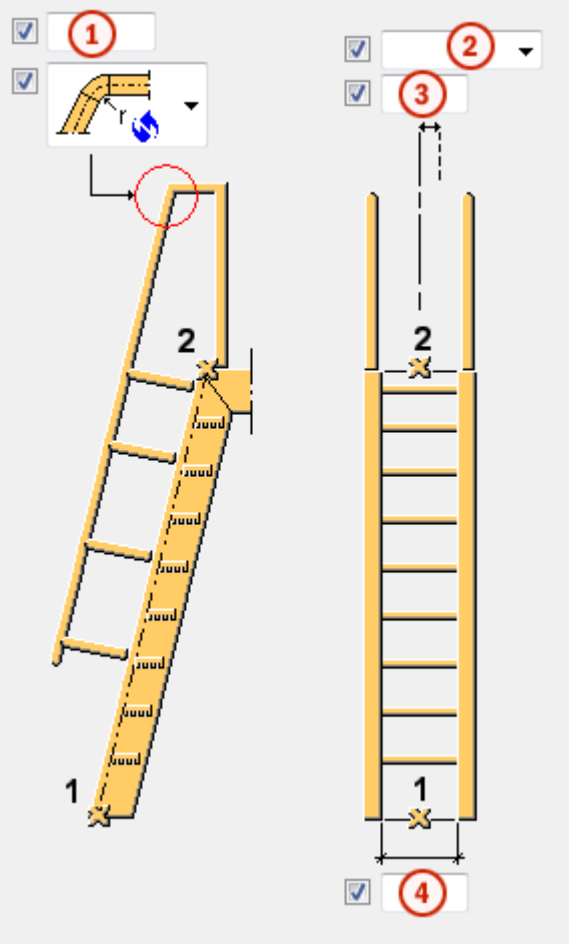


Use **Detail C** to connect the down stringer end to an end detail.



	Description	Default
1	Define whether the stringers are connected to an end detail.	`None`
2	Define the detail by selecting it from the component catalog.	
3	Select an attribute file for the detail.	`standard`

Ship ladder options

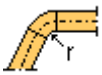
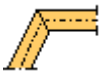
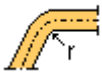


	Description	Default
1	Handrail elbow radius. This option is not active for **Type 4**.	`100 mm`
2	Define how to measure the dimension for the horizontal offset of the ship ladder. The options are: • **Left** To the left from the line defined by the picked points. • **Middle** The line defined by the picked points is the center line. • **Right** To the right from the line defined by the picked points.	`Middle`

	Description	Default
3	Horizontal offset. This option is not active for **Default** or **Middle**.	0 mm
4	Horizontal distance between stringers.	1000 mm

Handrail elbow bending cuts

This option is not active for **Type 4**.

Option	Description
	Default Bent elbow AutoDefaults can change this option.
	Bent elbow Separate elbow part is between rails.
	Fitting Rails are fitted.
	Bent rail Rail is bent.
	Separate rails Rails are not fitted.

Parts tab

Use the **Parts** tab to control the properties of the created parts.

Part properties

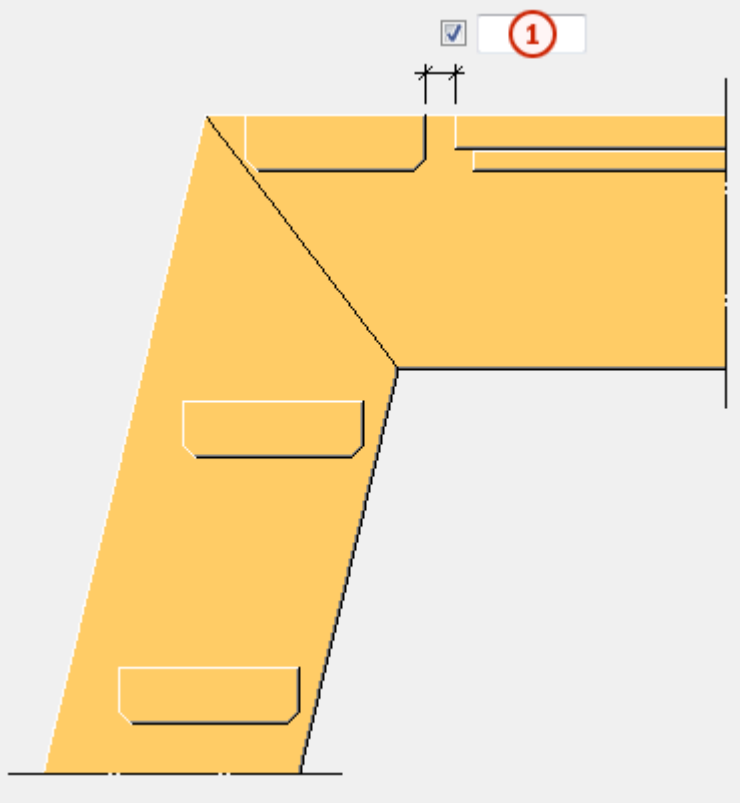
Option	Description	Default
Elbow **Handrail** **Middle rail** **Stanchion**	Define the elbow profile by selecting it from the profile catalog.	PD40*2
Plate in Detail B	Thickness of the plate.	5 mm
Support	Thickness of the platform support.	5 mm
Step	Select whether to use catalogue steps or	

Option	Description	Default
	profiles or to create the steps. Select the step from the list of catalogue steps or from the profile catalog. If you change your Tekla Structures environment, update the step list.	
Stringer	Define the stringer profile by selecting it from the profile catalog.	C200*100*5
Platform	Thickness of the platform.	50 mm
Bracket	Define the bracket profile by selecting it from the profile catalog.	

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

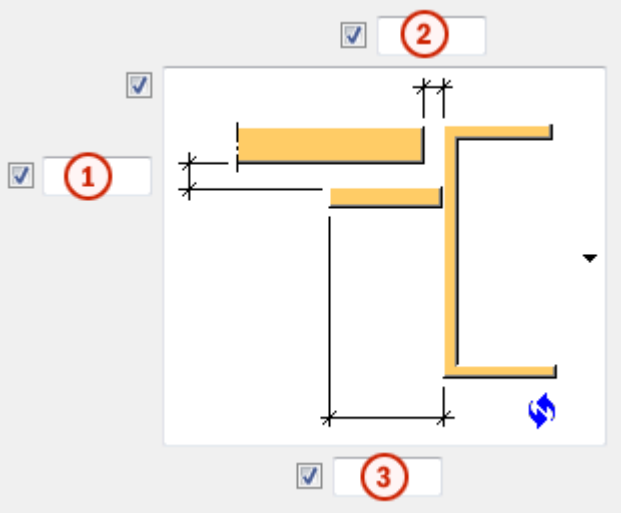
Platform tab
Use the **Platform** tab to control the platform offset and support.

Platform offset



	Description	Default
1	Platform horizontal offset from the top step. If the top step is not created, the platform horizontal offset is defined from the up nosing line top point.	0 mm

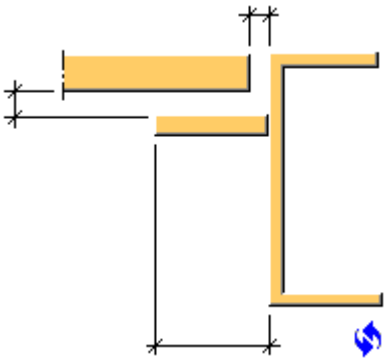
Platform offset and support

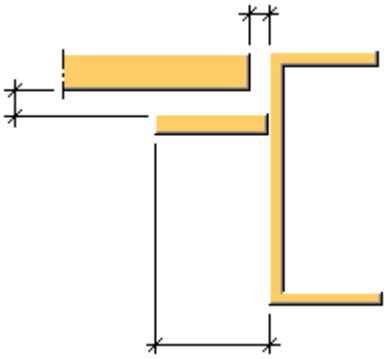
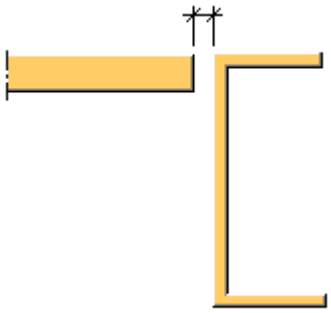


	Description	Default
1	Platform support vertical offset from the platform bottom.	0 mm
2	Platform horizontal offset from the stringer.	0 mm
3	Width of the platform support.	50 mm

Platform support creation

NOTE The top of the platform is always on the same level as the top of the stringer.

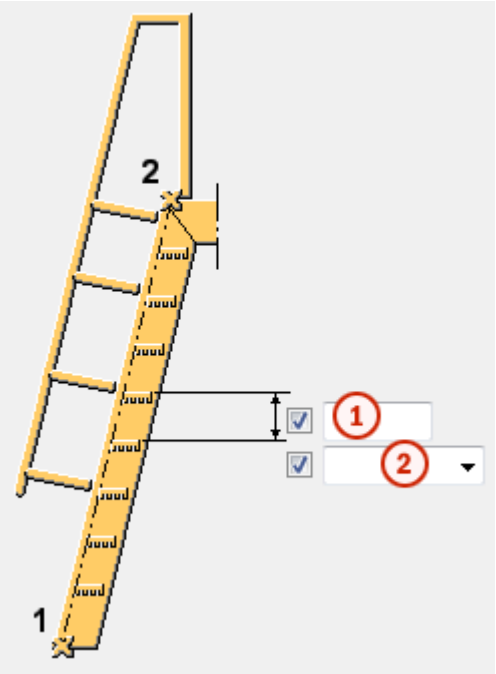
Option	Description
	Default Platform support is created. AutoDefaults can change this option.

Option	Description
	Platform support is created.
	Platform support is not created.

Steps tab

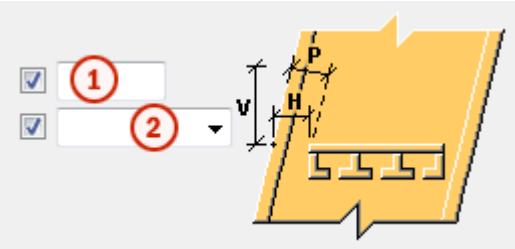
Use the **Steps** tab to control the step spacing and offset.

Step spacing



	Description	Default
1	Vertical spacing between the steps. The spacing depends on the step spacing type.	`300 mm`
1	Step spacing type. The steps are spaced from the second picked point to the first picked point **Equal** represents the maximum distance between the steps. **Exact** represents the exact distance between the steps.	`Equal`

Step offset



	Description	Default
1	Step offset from the stringers.	`0 mm`
2	Type of the step offset.	`Horizontal (H)`

Rails tab

Use the **Rails** tab to control the handrail and middle rail properties.

Middle rail and Handrail

Option	Description	Default
Middle rail to stanchion **Handrail to stanchion**	Define how the middle rails or handrails are connected to the stanchions.	Middle rails = `Connection` Handrails = `Weld`
Connection number	Define a connection that connects the rails to stanchion by selecting it from the component catalog.	`Round tube (23)`
Connection properties	Select an attribute file for the connection.	`standard`

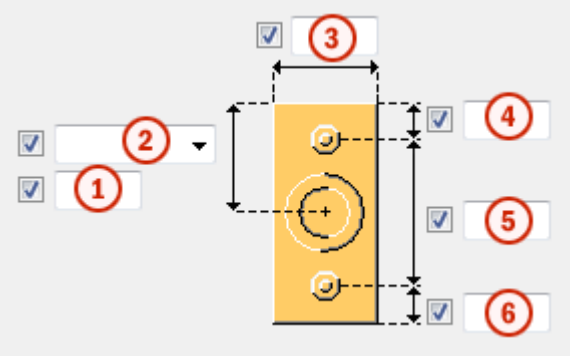
Handrail options

Option	Description	Default
Handrail max. length	Maximum length of the handrail.	`3000 mm`
Handrail cuts	Define how the handrails are cut. • **At max. stanchion** Starting from the handrail end, the **Handrail max. length** is measured along the handrail and the cut is made at the last stanchion included in the measured distance. • **At max. length** Starting from the handrail end, the **Handrail max. length** is measured along the rail and the cut is made at the maximum length. • **Every stanchion** A cut is made to the handrail at each stanchion.	`At max. stanchion`

Detail B tab

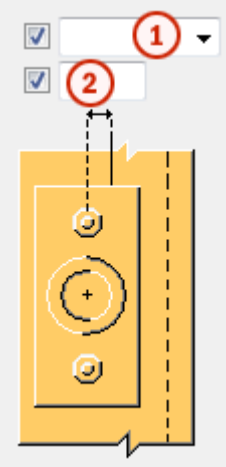
Use the **Detail B** tab to control the bolt properties on a plate that connects stanchions to stringers.

Vertical offset



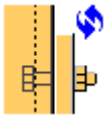
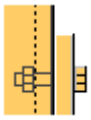
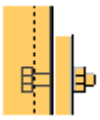
	Description
1	Plate vertical offset.
2	Define how to measure the dimension for the plate vertical offset. • **Top** From the top of the plate to the stanchion center. • **Middle** From the horizontal center line of the plate to the stanchion center. • **Below** From the bottom of the plate to the stanchion center.
3	Plate width.
4	Distance between top bolt and plate top.
5	Distance between the bolts.
6	Distance between bottom bolt and plate bottom.

Horizontal offset



	Description
1	Define how to measure the dimension for the horizontal plate offset. • **Left** • From the left of the plate to the stanchion center. • **Middle** From the horizontal center line of the plate to the stanchion center. • **Right** • From the right of the plate to the stanchion center.
2	Horizontal plate offset.

Bolting direction

Option	Description
	Default From plate to stringer AutoDefaults can change this option.
	From stringer to plate
	From plate to stringer

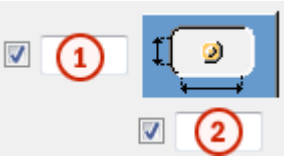
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

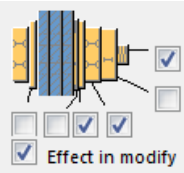
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Welds tab

Click the link below to find out more:

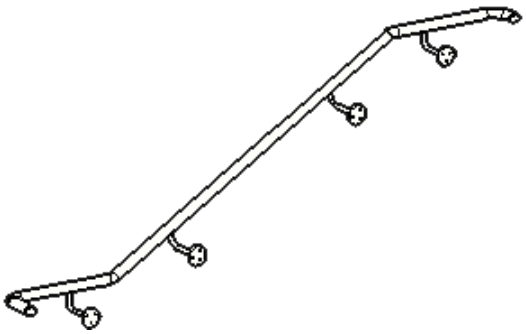
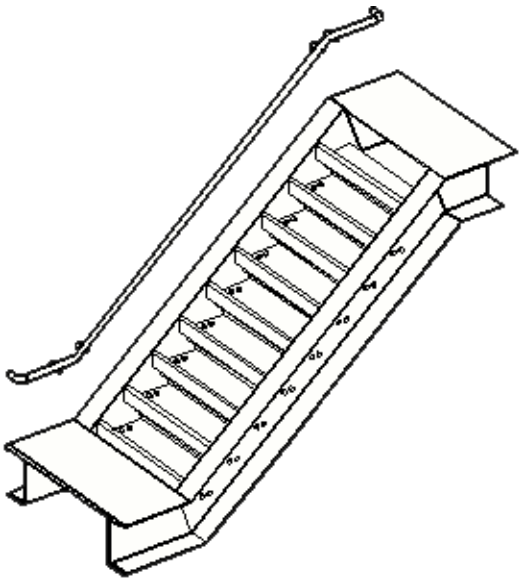
Wall Rails

Wall rails creates a horizontal or a sloped handrail with supporting elements. The handrail is connected to a wall.

Objects created

- Rail
- Elbows
- Endings
- Bracket arm
- Bracket base
- Bolts
- Welds

Use for

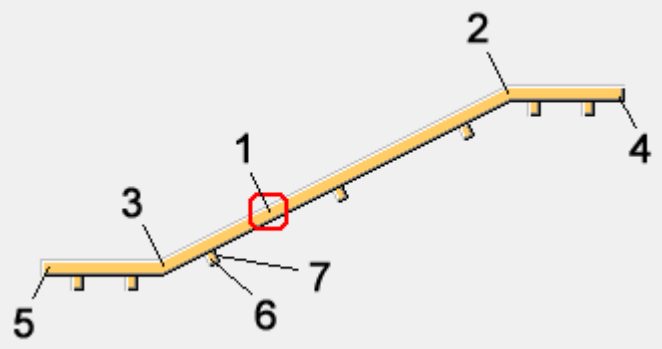
Situation	Description
	Rail with elbows, endings, and brackets.
	Rail on one side of a ladder, with elbows, endings, and brackets.
	Straight rail with brackets.

Selection order

1. Pick the start point
2. Pick the end point.

   The rail is created automatically.

Part identification key



	Part
1	Rail
2	Right elbow
3	Left elbow
4	Right ending
5	Left ending
6	Bracket arm
7	Bracket base

General tab

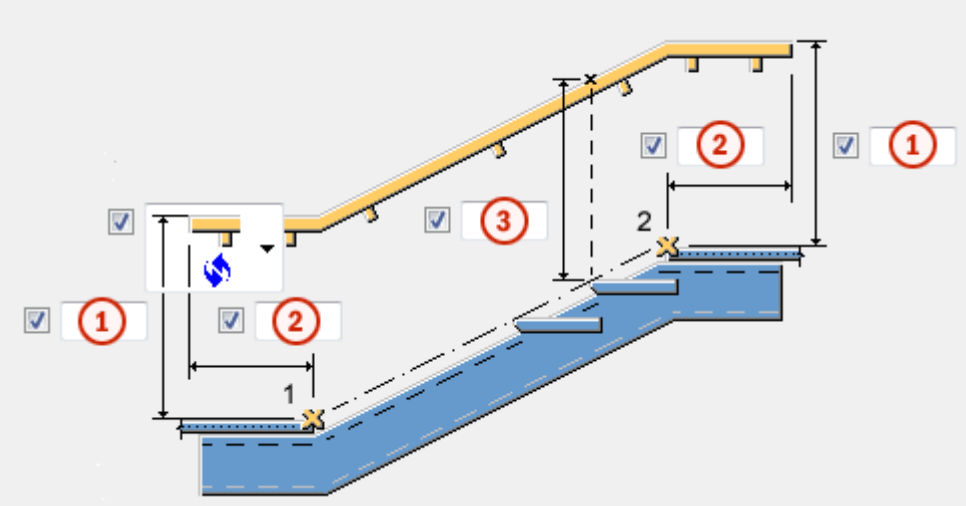
Use the **General** tab to control the assembly creation, rail dimensions and the rail offset.

Assembly creation

Option	Description	Default
Create Assembly	Define which parts form an assembly. The options are: • **All** All parts are included in an assembly, including brackets. Rail is the assembly main part.	`All`

Option	Description	Default
	• **Rail** Rail and elbows form an assembly. The sloped part of the rail is the assembly main part. • **No** No assembly is created.	

Rail dimensions



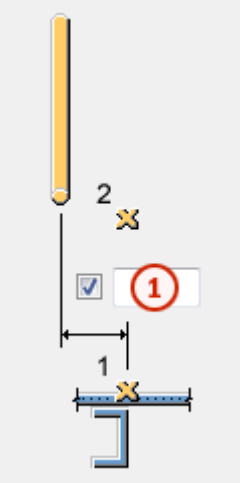
	Description	Default
1	Vertical distance between the rail and the start/end point.	0 mm
2	Horizontal distance from the rail's rightmost/leftmost edge to the start/end point. If you have created endings, the distance is measured from the endings' outmost edge.	0 mm
3	Vertical distance between the rail and the noseline.	0 mm

Rail ending dimensions

Define whether the vertical distance between the start/end point and the rail is measured from the top or the middle of the rail.

Option	Description
	Default From top AutoDefaults can change this option.
	From top
	From middle

Rail offset

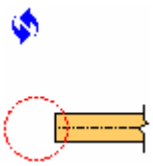
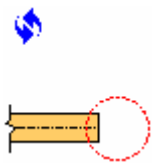
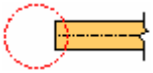
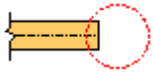


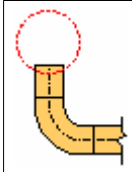
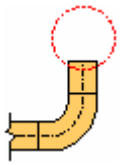
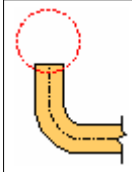
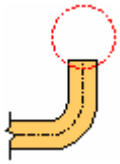
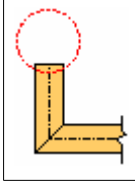
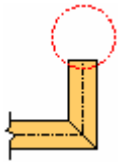
	Description	Default
1	Rail horizontal offset from the start/end point.	0 mm

Endings tab

Use the **Endings** tab to control the rail ending types and dimensions.

Ending types

Option	Option	Description
		Default No ending AutoDefaults can change this option.
		No ending

Option	Option	Description
		Separate bent elbow part between rails.
		Rail is bent.
		Rail and ending are fitted.

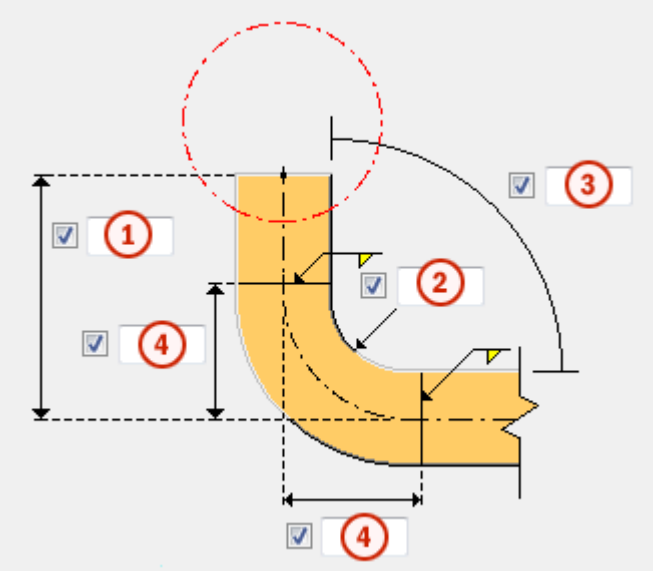
End detail

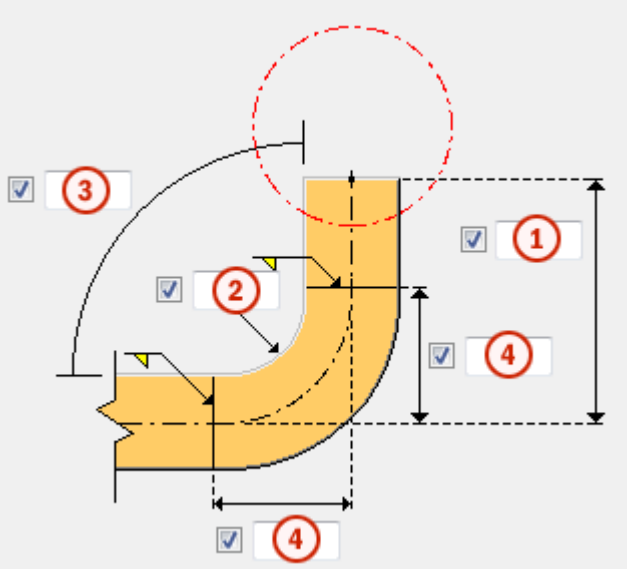
Option	Description	Default
End detail	Define a detail that is created to the ending by selecting it from the component catalog.	`None`
Attribute	Select an attribute file for the end detail.	`standard`

NOTE Some details do not work correctly with the bent rail. In that case you need to

- select another type of detail
- use some other elbow or ending type than bent rail.
- select the **Internal** bracket type.

Ending dimensions



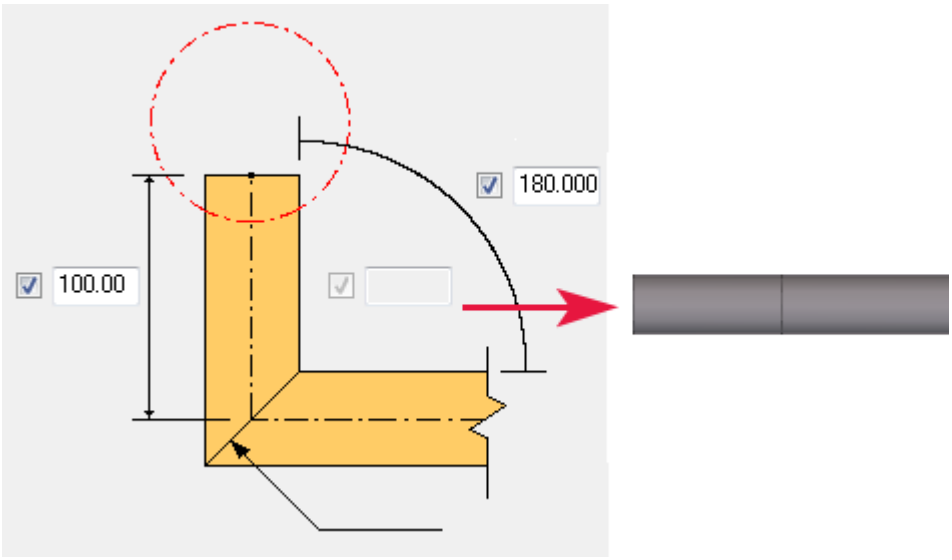


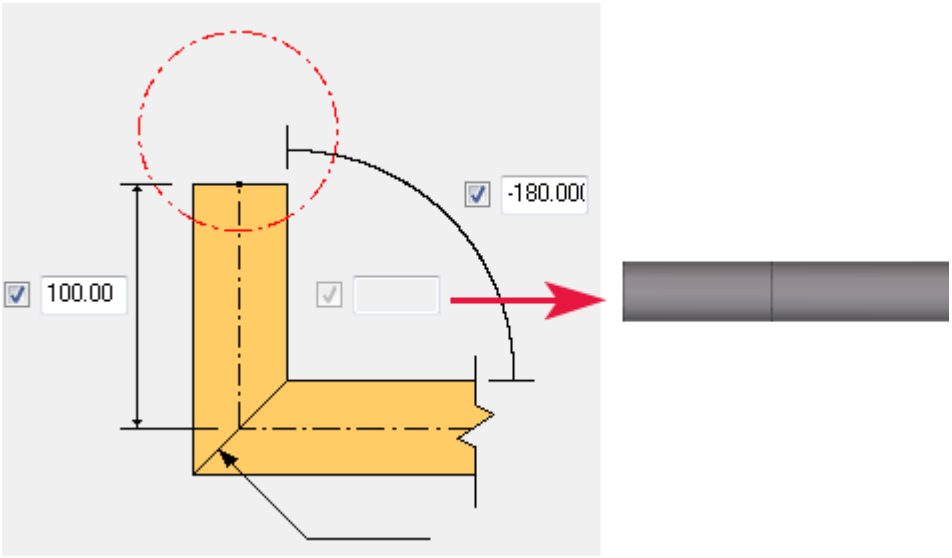
	Description	Default
1	Length of the straight part of the ending.	`100 mm`
2	Inner radius of the bent ending.	`30 mm`
3	Bent angle by entering a value between +90 and +180 degrees or -90 and -180 degrees.	`90 degrees`
4	Length of the bending.	

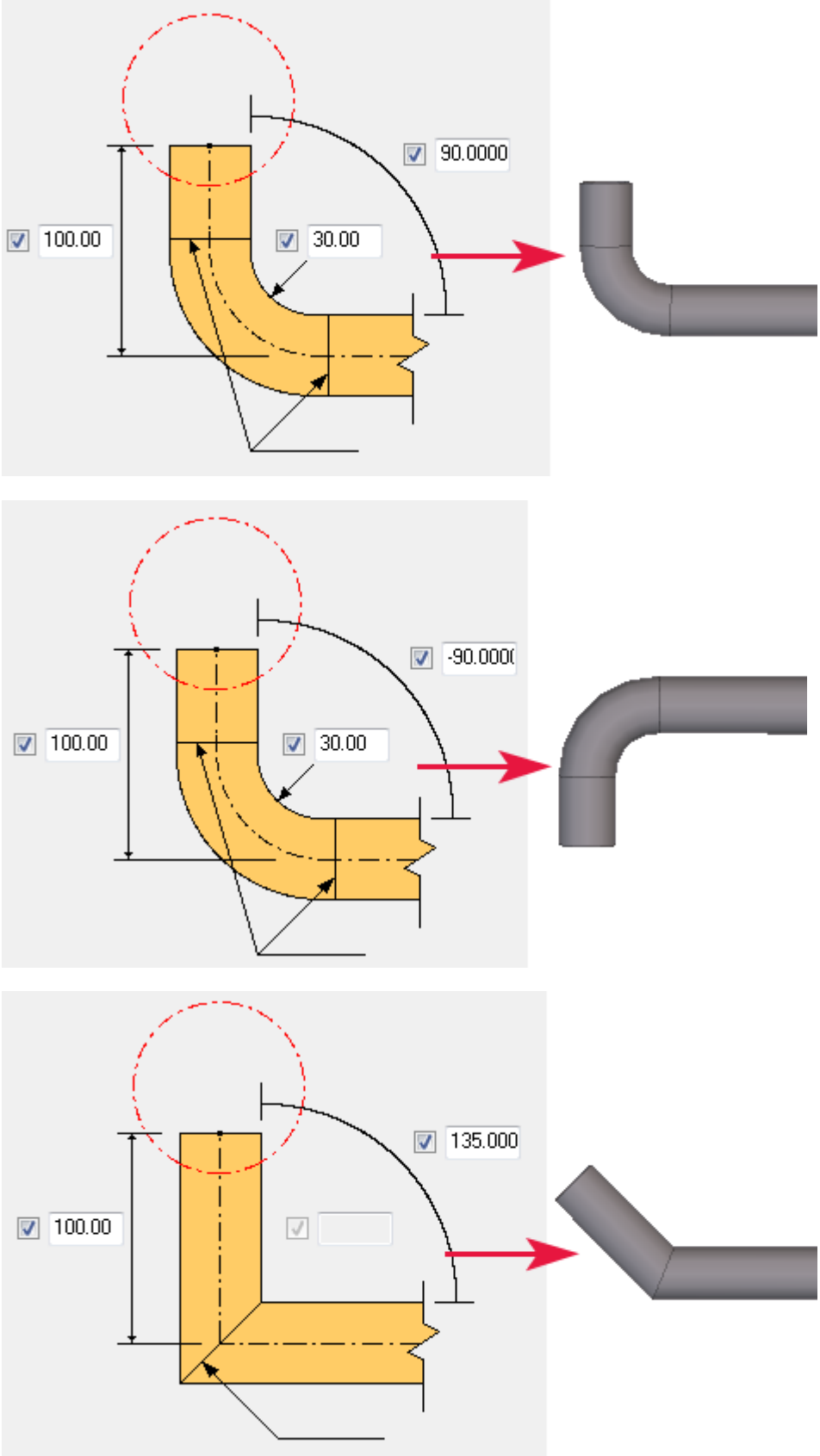
Rail orientation

Define the orientation of the handrail.

Bent angle examples



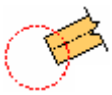
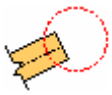
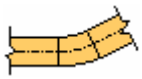
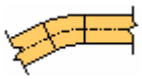
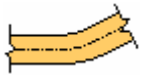
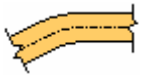
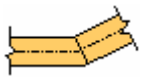
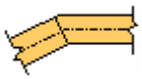
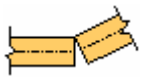
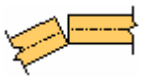



90.0000
100.00
30.00
-90.000(
100.00
30.00
135.000
100.00

Elbows tab

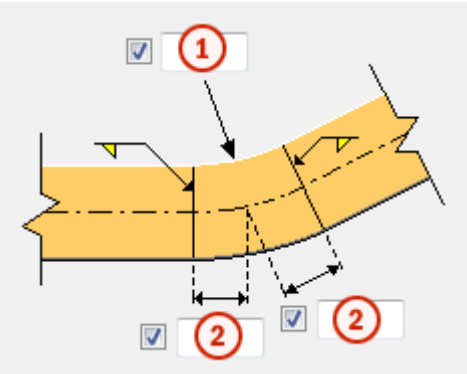
Use the **Elbows** tab to control the elbow types and dimensions.

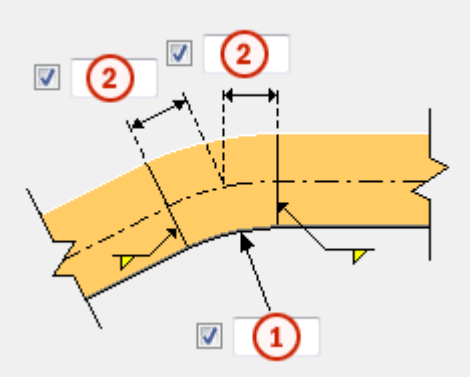
Elbow types

Option	Option	Description
		Default No elbow AutoDefaults can change this option.
		No elbow
		Separate bent elbow part between rails.
		Rail is bent.
		Rails are fitted.
		Rails are not fitted.

NOTE If you select some other option than **No elbow**, you need to enter a value for the horizontal distance from the rail's rightmost/leftmost edge to the start/end point on the **General** tab. Otherwise the rail is not created correctly.

Radius and length of bending





	Description	Default
1	Inner radius of the bent elbow.	`30 mm`
2	Length of the bending.	

Brackets tab

Use the **Brackets** tab to control the bracket types and dimensions.

Bracket options

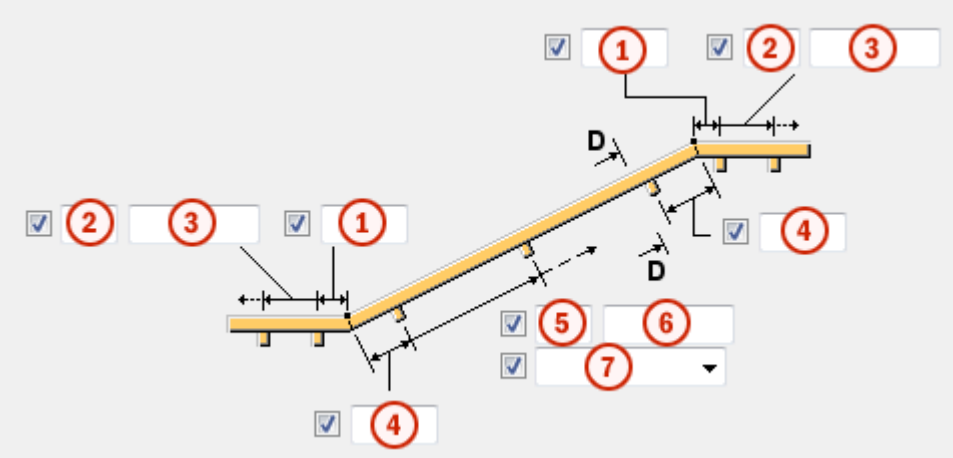
Option	Description	Default
Bracket type	Define how brackets are created. The options are: • **Internal** Brackets are created according to the profile selected on the **Parts** tab. • **Detail** Brackets are created according to the selected bracket detail. • **No** No brackets are created.	`Internal`
Bracket base	Define whether the bracket base is created or not.	`Yes`
Bracket detail	Define a system or a custom detail that is	

Option	Description	Default
	used as a bracket by selecting it from the component catalog. This field is active only if you have set the **Bracket type** to **Detail**.	
Attribute	Select an attribute file for the detail.	`standard`
Connect bracket detail to	Define the part to which the bracket detail is connected.	`Bracket arm`

Bracket direction

Option	Description
	Default Brackets on the left side AutoDefaults can change this option.
	Brackets on the left side
	Brackets on the right side

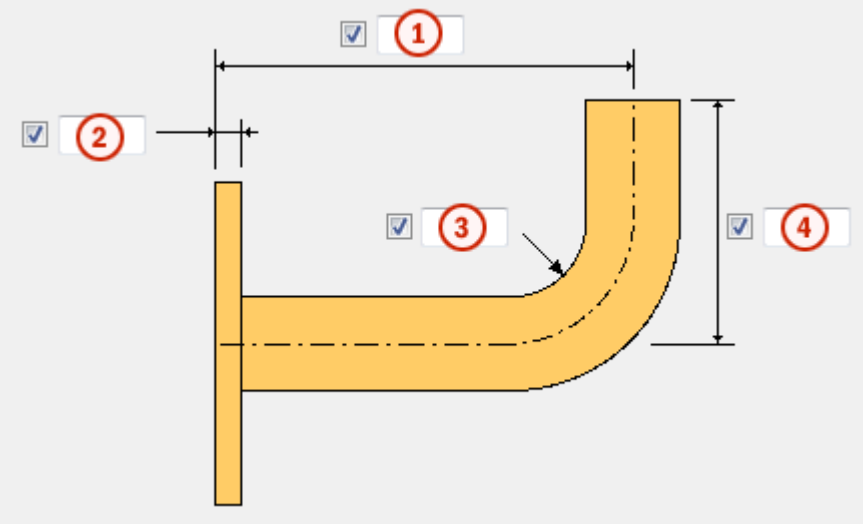
Bracket positioning



	Description	Default
1	Distance between the start/end point of the rail and the bracket.	`100 mm`
2	Number of brackets.	`0`

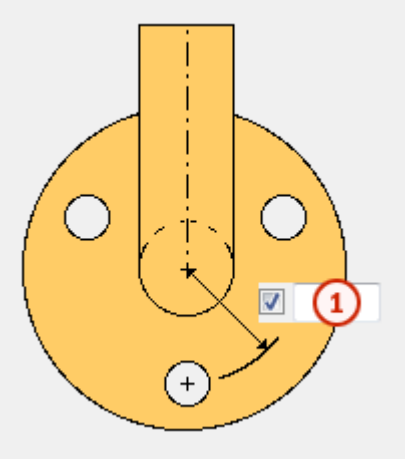
	Description	Default
3	Bracket spacing. Use a space to separate bracket spacing values.	0 mm
4	Distance between the start/end point of the rail and the bracket.	100 mm
5	Number of brackets.	0
6	Bracket spacing. Spacing depends on the selected bracket spacing type.	0 mm
7	Spacing type. If you select **Maximum**, the component creates the mininum amount of brackets needed without exceeding the entered spacing value. The amount of brackets is then equally distributed along the rail.	Exact

Bracket dimensions



	Description	Default
1	Bracket arm width. The dimension is measured from the bracket base.	120 mm
2	Bracket base thickness.	50 mm
3	Bent inner radius of the bracket arm.	10 mm
4	Bracket arm vertical length. The dimension is measured from the bent radius.	5 mm

Hole placement in bracket base

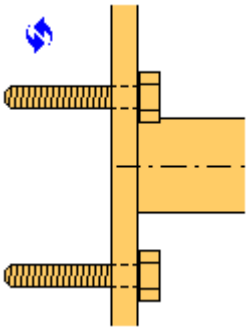
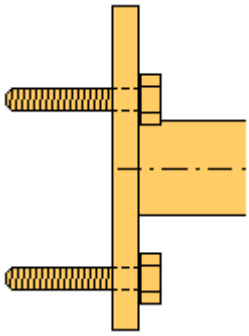


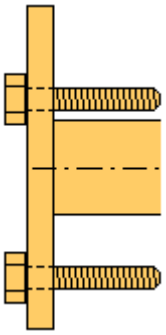
	Description	Default
1	Radius of the circle on which the bracket holes are placed.	17 mm

Bolts tab

Use the **Bolts** tab to control the bolts that connect the bracket base to a wall.

Bolting direction

Option	Description
	Default From bracket to wall AutoDefaults can change this option.
	From bracket to wall

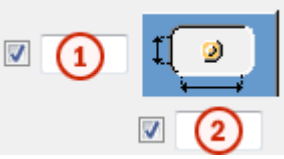
Option	Description
	From wall to bracket

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.

Option	Description	Default
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

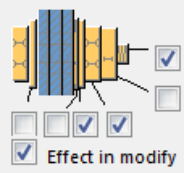
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Parts tab

Use the **Parts** tab to control dimensions of the created parts.

Part guide

Select a part from the list. The selected part is displayed.

Part dimensions

Option	Description	Default
Rail	Define a profile by selecting it from the profile catalog.	`CHS40*3`
Right elbow	Define a profile by selecting it from the profile catalog.	`Not created`
Left elbow	Define a profile by selecting it from the profile catalog.	`Not created`
Right ending	Define a profile by selecting it from the profile catalog.	`Not created`
Left ending	Define a profile by selecting it from the profile catalog.	`Not created`
Bracket arm	Define a profile by selecting it from the profile catalog.	`D12`
Bracket base	Define a profile by selecting it from the profile catalog.	`D60`

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Welds tab

Click the link below to find out more:

5.14 Stiffeners and gussets

This section introduces components that can be used in steel stiffeners and gussets.

- Beam with stiffener (129) (page 2358)
- Gusset stiffeners (171) (page 2382)
- Column with stiffeners W (182) (page 2386)
- Column with stiffeners (186) (page 2418)
- Column with stiffeners S (187) (page 2447)
- Column with stiffeners (188) (page 2474)
- Stiffeners (1003) (page 2505)
- Multiple stiffeners (1064) (page 2509)
- Standard gusset (1065) (page 2515)

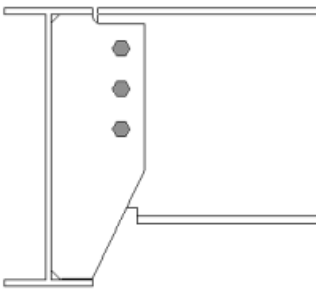
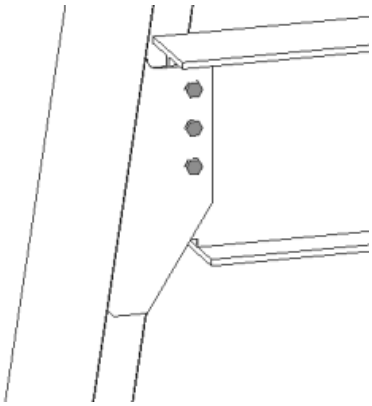
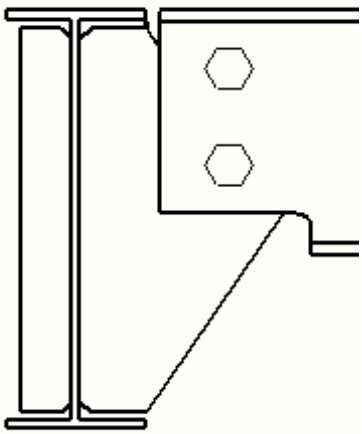
Beam with stiffener (129)

Beam with stiffener (129) connects a beam to another beam with a bolted and welded shear tab. The connection can be used at the back of a U-profile. The secondary beam can be leveled or sloped and/or skewed.

Objects created

- Shear tabs (1 or 2)
- Stiffener (optional)
- Haunch plates (optional)
- Welds
- Bolts
- Cuts

Use for

Situation	Description
	Full depth shear tab.
	Full depth shear tab. The secondary part is sloped and/or skewed.
	Shear tab with a beam stiffener.

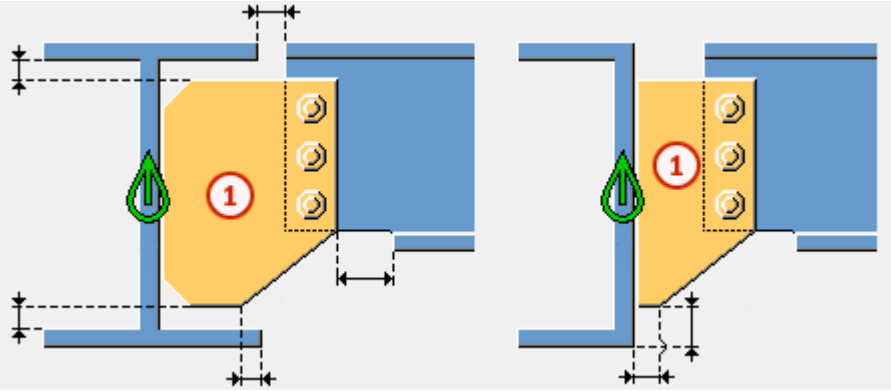
Selection order

1. Select the main part (beam).

2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key



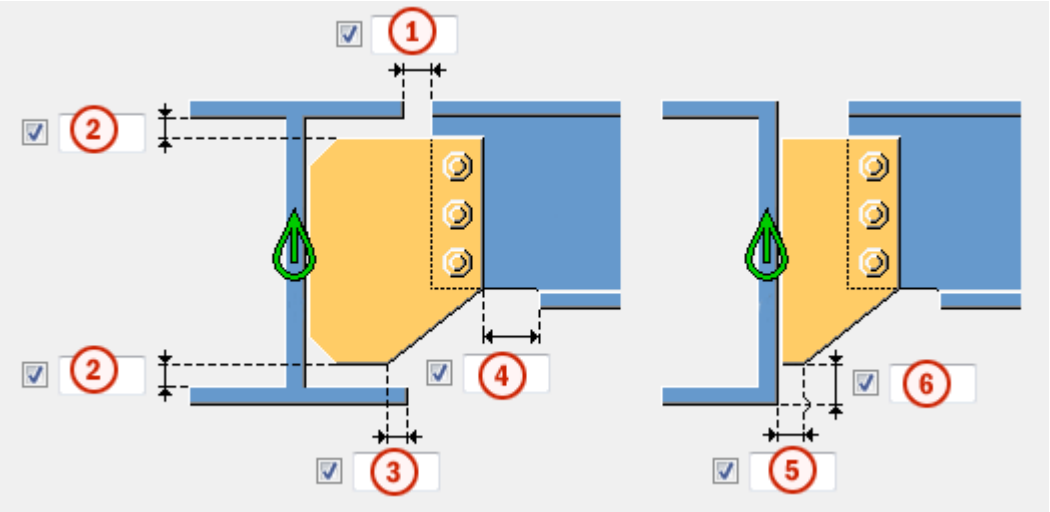
	Part
1	Shear tab

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to control the position of the shear tab, and the beam flange and the web cuts.

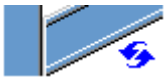
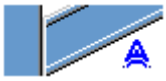
Dimensions



	Description	Default
1	Cut of the secondary part. Cutting the secondary part creates a gap between the main part and the secondary part.	10 mm
2	Shear tab edge distance from the main part flange edge.	0
3	Distance from the corner of the shear tab to the edge of the main part flange.	
4	Size of the strip made to the secondary part flange. The cut of the flange is defined from the shear tab edge.	The flange is automatically stripped when the shear tab crosses the flange. 20 mm
5	Distance from the edge of the main part to the corner of the shear tab.	20 mm
6	Distance from the bottom edge of the main part to the bottom edge of the shear tab.	10 mm

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.
	Square Cuts the end of the secondary beam square.
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.

Option	Description
	Square cut closer to the main part web Cuts the end of the secondary beam square and places the beam closer to the main part web.
	Clipped flange Cuts the corner of the flange at the end of the secondary beam.

Beam flange cut

Define how the secondary beam flange end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the flange bevel.
	Square Cuts a part of the flange square and leaves a part of it bevel.

Beam web cut

Define how the secondary beam web end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the web bevel when the end of the secondary beam is cut bevel.
	Square Cuts the end of the web square even if the end of the secondary beam is cut bevel.

Beam bottom flange cut

Option	Description
	Default Notch Define the notch dimensions. AutoDefaults can change this option.
	Notch Define the notch dimensions. The bottom of the secondary beam is notched if the shear tab crosses the flange.
	Flange cut The secondary beam flange is cut on the same side as the shear tab if the shear tab crosses the flange.

Plates tab

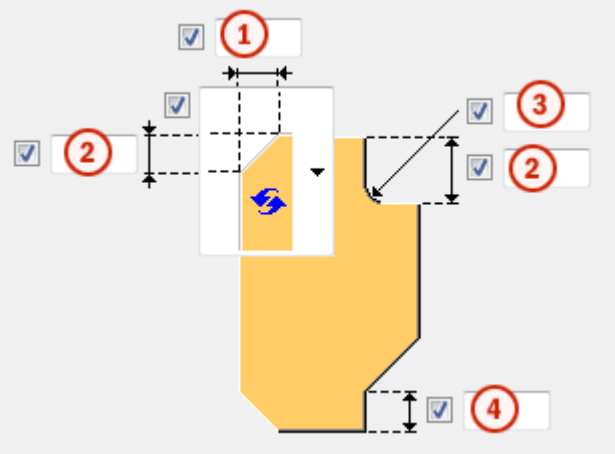
Use the **Plates** tab to control the size, position, number, and shape of the shear tab.

Shear tab

Option	Description
Tab plate	Shear plate tab thickness and width.

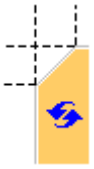
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Shear tab chamfers



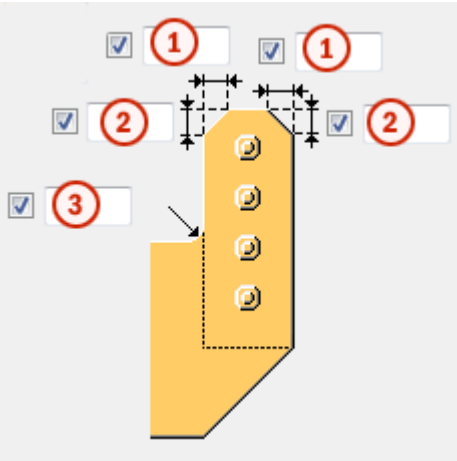
	Description
1	Horizontal dimension of the shear tab chamfer.
2	Vertical dimension of the shear tab chamfer.
3	Vertical and the horizontal dimension of the shear tab chamfer.
4	Vertical dimension from the bottom edge of the shear tab to the lower shear tab corner.

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer

Option	Description
	Concave arc chamfer

Inner shear tab chamfers



	Description
1	Horizontal dimension of the shear tab chamfer.
2	Vertical dimension of the shear tab chamfer.
3	Radius and the vertical dimensions of the shear tab inner chamfer.

Chamfer type

Option	Option	Description
		Default No chamfer AutoDefaults can change this option.
		No chamfer
		Line chamfer
		Convex arc chamfer

Option	Option	Description
		Concave arc chamfer

Inner chamfer type

Option	Description
	Default Concave arc chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

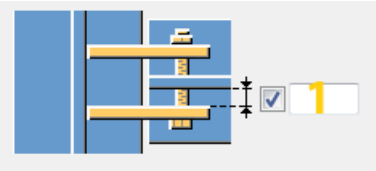
Shear tab position

Define the number and the side of shear tabs in single shear tab connections.

Option	Description
	Default Far side shear tab AutoDefaults can change this option.
	Automatic The component automatically selects either the near side or the far side shear tab. The tab is created to the side of the secondary part when the angle between the main part and the secondary part is less than 90 degrees.
	Far side shear tab

Option	Description
	Near side and far side shear tab
	Near side shear tab

Gap between shear tabs



	Description	Default
1	Gap between the secondary part web and shear tab. This only affects connections with two shear tabs.	0

Shear tab orientation

Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Sloped The shear tab is sloped in the direction of the secondary beam. Both vertical edges of the shear tab are cut parallel to the end of the secondary beam.
	Square

Stiffeners tab

Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position, and type.

Opposite web stiffener plate dimensions

Option	Description
Opposite web stiffener	Opposite web stiffener plate thickness, width and height.

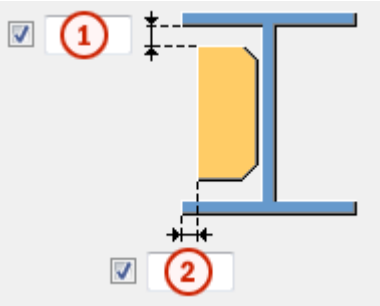
Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Stiffener creation

Option	Description
	Default No stiffeners are created. AutoDefaults can change this option.
	Full Creates a full stiffener of the same height as the web of the main part.
	Determined by shear tab Tekla Structures determines the size of the stiffener based on the shear tab size. Tekla Structures attempts to keep the bottom edges of the stiffener plate and shear tab level, if possible.

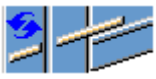
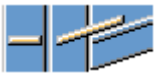
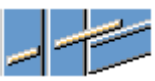
Option	Description
	Partial Leaves a gap between the stiffener plate and the bottom flange of the main part.
	No stiffeners are created.

Stiffener gap

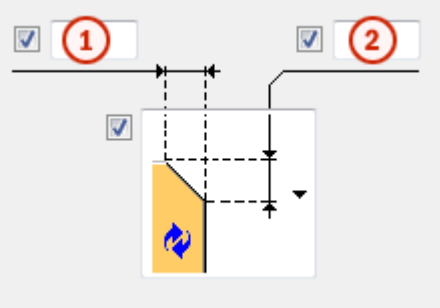


	Description
1	Size of the gap between the main part flange and the stiffener.
2	Distance from the edge of the main part flange to the edge of the stiffener.

Stiffener orientation

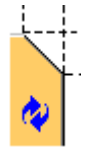
Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are perpendicular to the main part.
	Stiffeners are parallel to the secondary part.

Chamfer dimensions



	Description
1	Horizontal dimension of the chamfer.
2	Vertical dimension of the chamfer.

Chamfer type

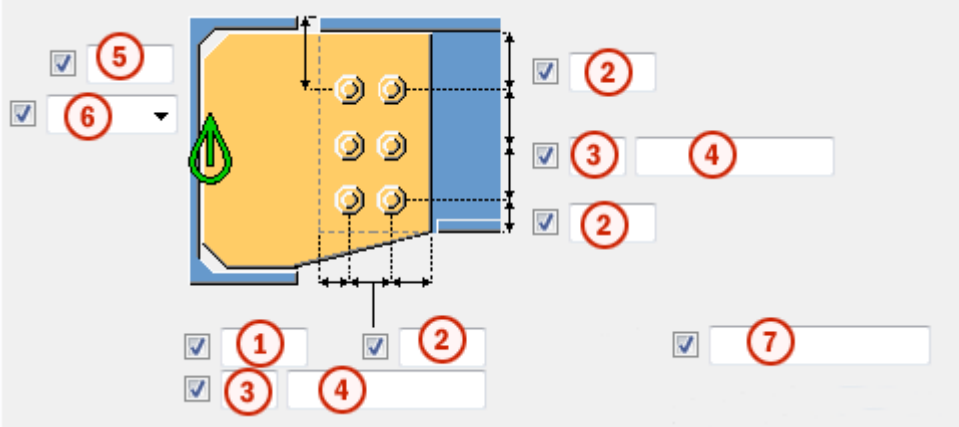
Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

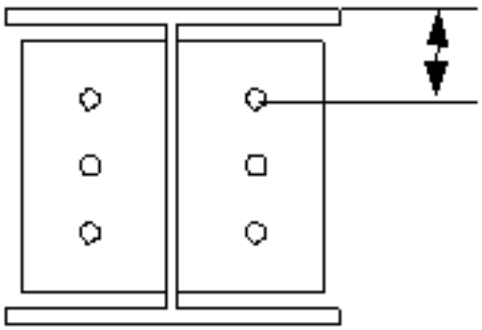
Bolts tab

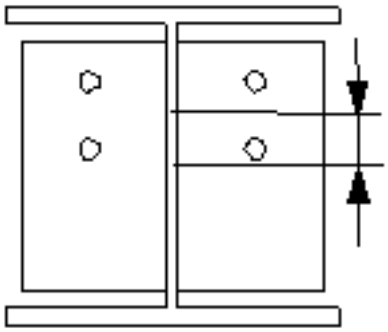
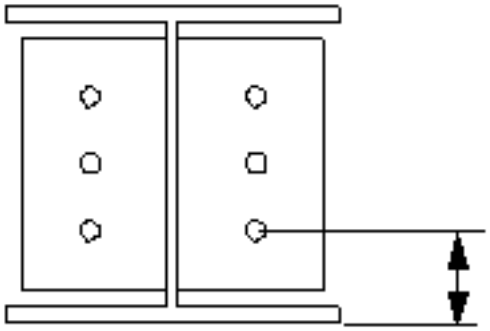
Use the **Bolts** tab to control the properties of the bolts that connect the shear tab to the secondary part.

Bolt group dimensions

Bolt group dimensions affect the size and shape of the shear tab.

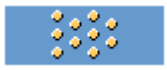


	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position.
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1

Option	Description
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt group orientation

Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Staggered Bolts are staggered in the direction of the secondary part.
	Square Square bolt group is positioned horizontally.
	Sloped Square bolt group is sloped in the direction of the secondary part.

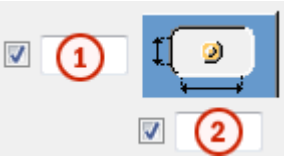
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

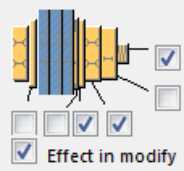


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Notch tab

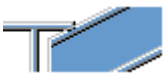
Use the **Notch** tab to automatically create notches for the secondary beams and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

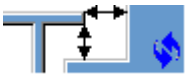
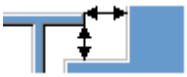
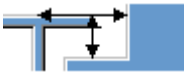
Automatic notching options affect both the top and the bottom flange.

Notch shape

Automatic notching is switched on when you select a notch shape.

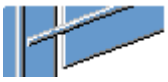
Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

Enter the horizontal and vertical values for the cuts.

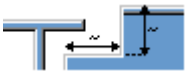
Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.

Option	Description
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.

Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.

Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

Side of flange notch

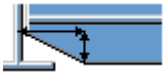
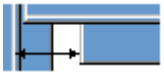
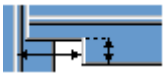
The side of flange notch defines on which side of the beam the notches are created.

Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

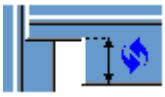
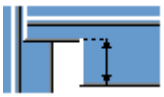
Flange notch shape

The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.

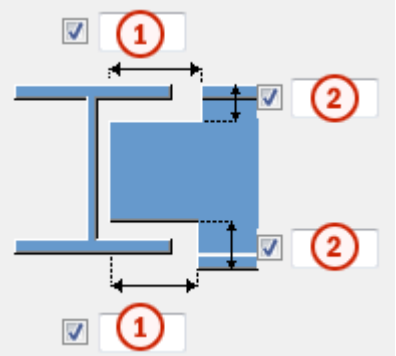
Option	Description
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.

Cut dimensions



	Description	Default
1	Dimensions for the horizontal flange cuts.	10 mm
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest 5 mm.

Haunch tab

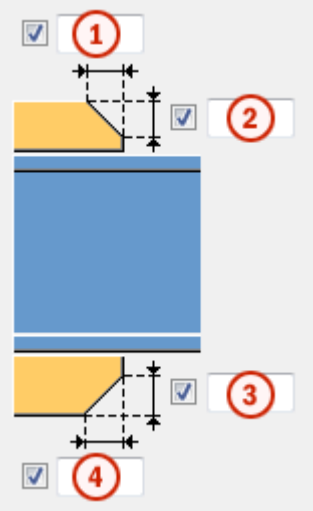
Use the **Haunch** tab to control the haunch plate creation and chamfers in the secondary beam flanges.

Haunch plates

Option	Description
Top plate	Top haunch plate thickness, width and height.
Bottom plate	Bottom haunch plate thickness, width and height.

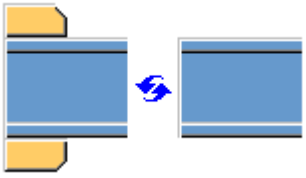
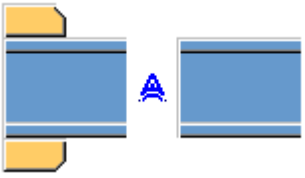
Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Haunch plate chamfers



	Description
1	Width of the top haunch plate chamfer.
2	Height of the top haunch plate chamfer.
3	Height of the bottom haunch plate chamfer.
4	Width of the bottom haunch plate chamfer.

Haunch plate creation

Option	Description
	Default Top and bottom haunch plates are created, if needed. AutoDefaults can change this option.
	Automatic Top or bottom haunch plate or both are created, if needed.
	Top and bottom haunch plates are created. To create a single plate, enter `0` in the thickness (**t**) field for the plate you do not need (top or bottom plate).

Option	Description
	Haunch plates are not created.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

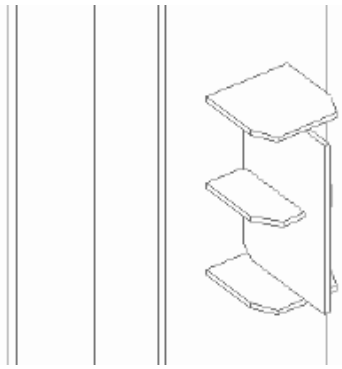
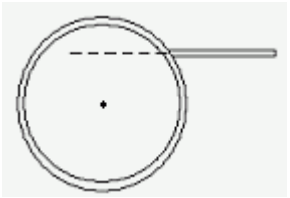
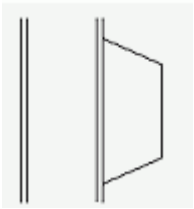
Gusset stiffeners (171)

Gusset stiffeners (171) creates 2 or 3 (default) stiffener plates and welds them to an existing gusset plate and a beam or a column. You can also create chamfers to the gusset plate and stiffeners.

Objects created

- Stiffener plates
- Welds
- Cuts

Use for

Situation	Description
	3 stiffeners welded to column flange and gusset plate.
	Gusset stiffener that is not perpendicular to an I-beam.
	Gusset stiffener for a pipe or bar. Notice that the stiffener does not have to be located on the centerline of the pipe or bar.
	Chamfered gusset stiffener.

Before you start

Create a gusset plate, and a beam or a column. Use I, C, tube, or RHS profiles.

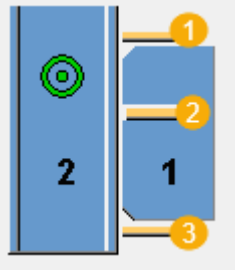
TIP To create a gusset plate, use the **Standard gusset (1065)** component or the plate command that creates a contour plate.

Selection order

1. Select the main part (gusset plate).
2. Select the secondary part (beam or column).

   The connection is created automatically when the secondary part is selected.

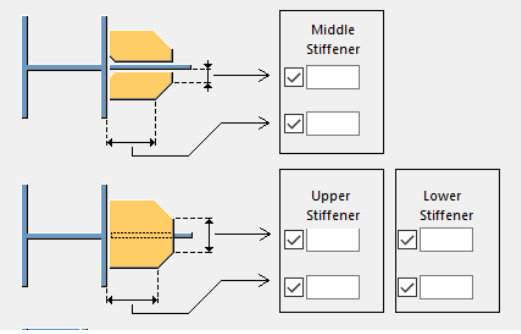
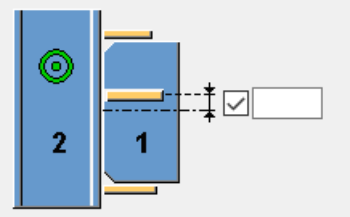
Part identification key



	Description
1	Upper stiffener
2	Middle stiffener
3	Lower stiffener

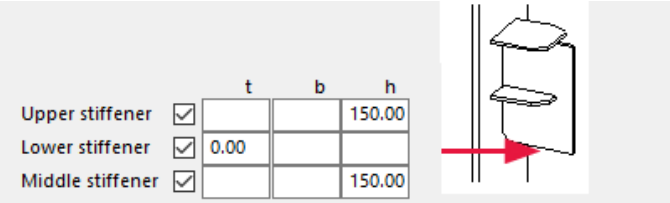
Picture tab

Use the **Picture** tab to define the location, shape, and dimensions of the stiffener plates.

Option	Description
	Define the horizontal and vertical dimensions of the stiffener plates.
	Define the middle stiffener offset from the gusset center line.
Stiffener shapes	Select the stiffener shape.
Flip stiffener directions	Select **Yes** to flip the stiffener direction.

Parts tab

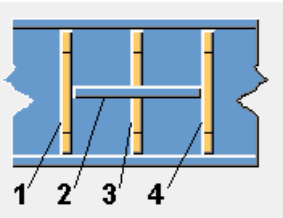
Use the **Parts** tab to define the stiffener properties.

	Description
Upper stiffener, **Lower stiffener**, **Middle stiffener**	Define the thickness, width and height of the stiffener. By default, 3 stiffeners are created. To remove a stiffener, enter 0 as the thickness for that stiffener. 

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Chamfer tab

Use the **Chamfer** tab to define the chamfer shapes and dimensions for the stiffeners and gusset plate.



General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

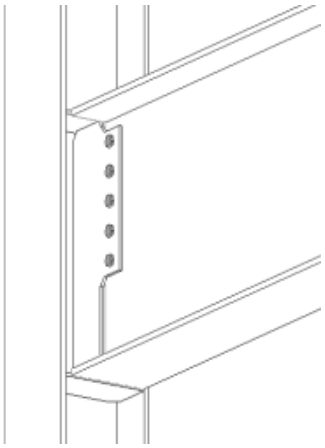
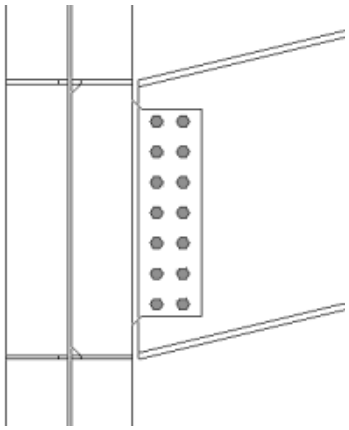
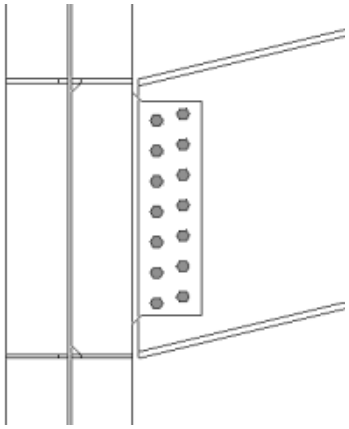
Column with stiffeners W (182)

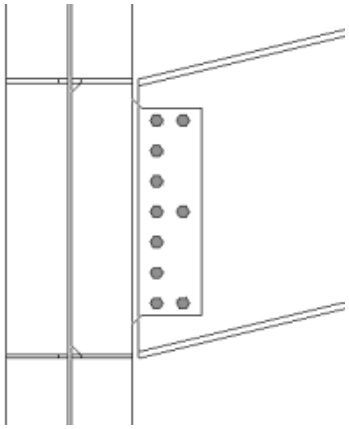
Column with stiffeners W (182) connects a column to a beam with a shear tab. The shear tab is welded to the main part and bolted to the secondary part web. The secondary beam can be leveled or sloped.

Objects created

- Shear tabs (1 or 2)
- Stiffeners (optional)
- Haunch plates (optional)
- Web doubler plate (optional)
- Weld backing bars (optional)
- Bolts
- Welds
- Cuts

Use for

Situation	Description
	Shaped shear tab with column stiffeners.
	Shaped shear tab with column stiffeners. The secondary part is sloped.
	Shaped shear tab with column stiffeners. Bolts are aligned with the secondary part.

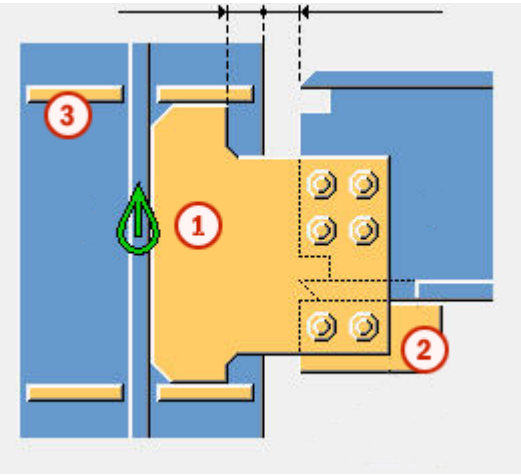
Situation	Description
	Shaped shear tab with column stiffeners. Bolts can be deleted.

Selection order

1. Select the main part (column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key



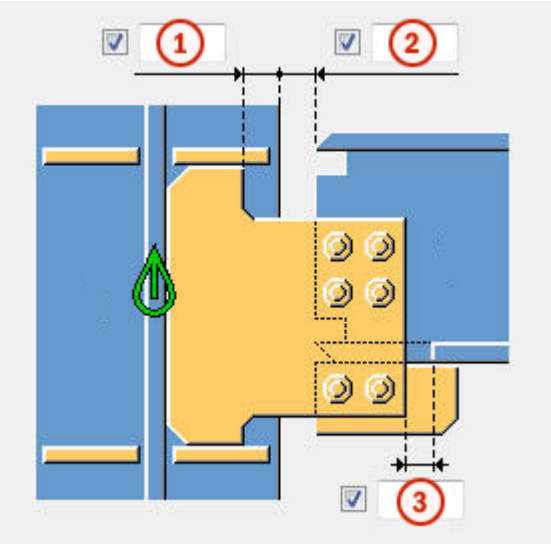
	Part
1	Shear tab
2	Haunch plate
3	Stiffener

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to control the position of the shear tab corner and to define how the beam end is cut.

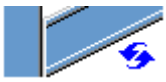
Dimensions



	Description	Default
1	Shear tab edge distance from the main part flange edge.	
2	Cut of the secondary part. Cutting the secondary part creates a gap between the main part and the secondary part.	20 mm
3	Size of the strip made to the secondary part flange. The cut of the flange is defined from the shear tab edge.	The flange is automatically stripped when the shear tab crosses the flange. 20 mm

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.

Option	Description
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.
	Square Cuts the end of the secondary beam square.
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.

Plates tab

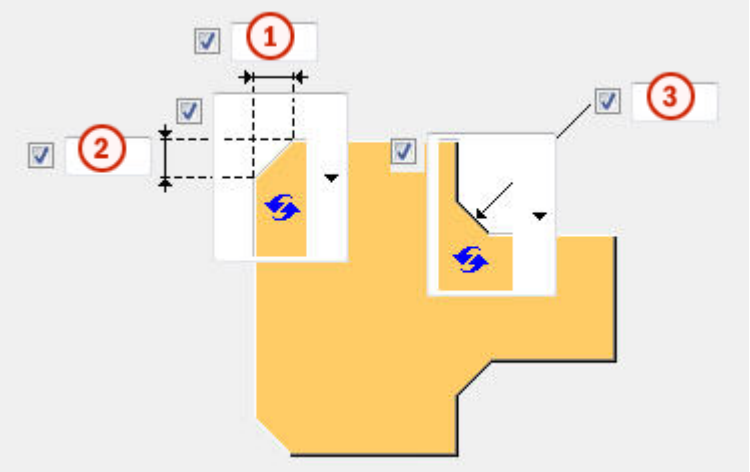
Use the **Plates** tab to control the size, position, number, orientation and shape of the shear tab.

Shear tab plate

Option	Description
Tab plate	Shear tab plate thickness, width and height.

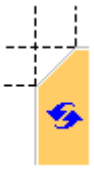
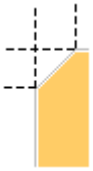
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Shear tab chamfers



	Description
1	Horizontal dimension of the shear tab chamfer.
2	Vertical dimension of the shear tab chamfer.
3	Vertical and the horizontal dimension of the shear tab chamfer.

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer

Option	Description
	Concave arc chamfer

Chamfer type dimensions

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Concave arc chamfer

Shear tab orientation

Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Sloped The shear tab is sloped in the direction of the secondary beam.

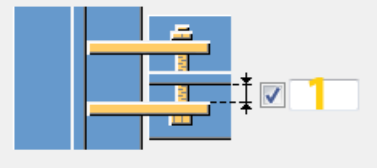
Option	Description
	Both vertical edges of the shear tab are cut parallel to the end of the secondary beam.
	Square

Shear tab position

Define the number and the side of shear tabs in single shear tab connections.

Option	Description
	Default Far side shear tab AutoDefaults can change this option.
	Automatic The component automatically selects either the near side or the far side shear tab. The tab is created to the side of the secondary part when the angle between the main part and the secondary part is less than 90 degrees.
	Far side shear tab
	Near side and far side shear tab
	Near side shear tab

Gap between shear tabs



	Description	Default
1	Gap between the secondary part web and shear tab. This only affects connections with two shear tabs.	0

Stiffeners tab

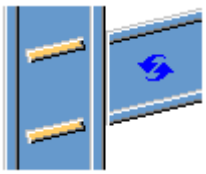
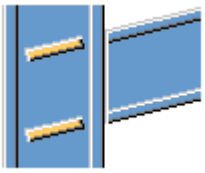
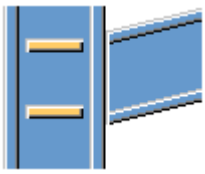
Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position and type.

Stiffener plate dimensions

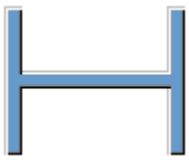
Option	Description
Top NS	Top near side stiffener thickness, width and height.
Top FS	Top far side stiffener thickness, width and height.
Bottom NS	Bottom near side stiffener thickness, width and height.
Bottom FS	Bottom far side stiffener thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Stiffener orientation

Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are parallel to the secondary part.
	Stiffeners are perpendicular to the main part.

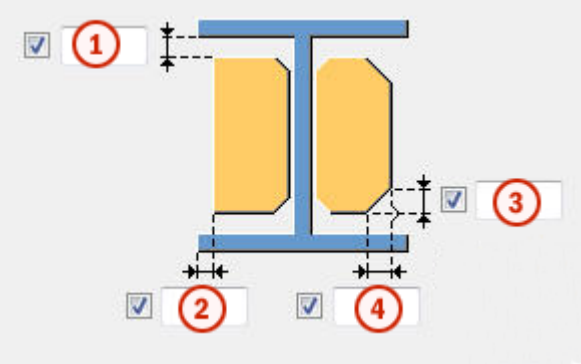
Stiffener creation

Option	Description
	Default Stiffeners are created. AutoDefaults can change this option.
	Automatic Stiffeners are created when necessary.
	No stiffeners are created.
	Stiffeners are created.

Stiffener shape

Option	Description
	Default Line chamfered stiffener plates AutoDefaults can change this option.
	Automatic Line chamfered stiffener plates
	Square stiffener plates Stiffener plates with a gap for the main part web rounding
	Line chamfered stiffener plates

Stiffener gap and chamfer size

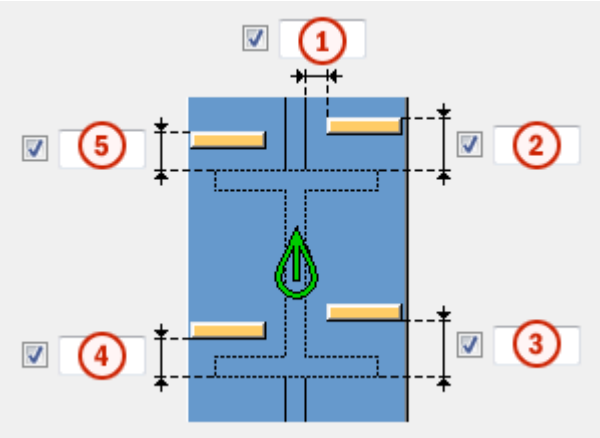


	Description
1	Size of the gap between the flanges and the stiffener.
2	Distance from the edge of the flange to the edge of the stiffener.
3	Vertical dimension of the stiffener line chamfer.
4	Horizontal dimension of the stiffener chamfer or radius of arc type chamfer.

Define the chamfer shape.

Option	Description
	Default No chamfers AutoDefaults can change this option.
	No chamfers
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

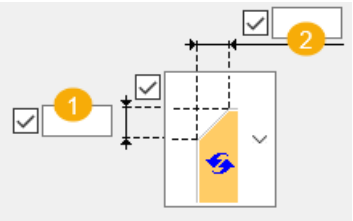
Stiffener positions



	Description
1	Size of the gap between the stiffener and the beam web edge.
2	Size of the gap between the top near side stiffener and the beam flange edge.
3	Size of the gap between the bottom near side stiffener and the beam flange edge.
4	Size of the gap between the bottom far side stiffener and the beam flange edge.

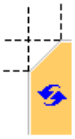
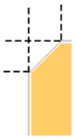
	Description
5	Size of the gap between the top far side stiffener and the beam flange edge.

Chamfer dimensions



	Description	Default
1	Vertical dimension of the chamfer.	`10 mm`
2	Horizontal dimension of the chamfer.	`10 mm`

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Option	Description
	Line and arc chamfer

Haunch

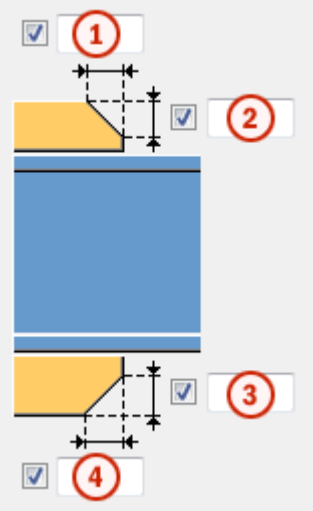
Use the **Haunch** tab to control the haunch plate creation and chamfers in the secondary beam flanges.

Haunch plates

Option	Description
Top plate	Top haunch plate thickness, width and height.
Bottom plate	Bottom haunch plate thickness, width and height.

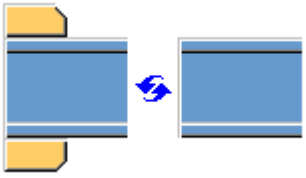
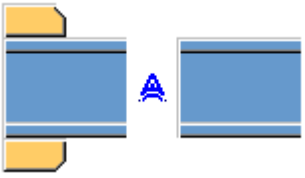
Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Haunch plate chamfers



	Description
1	Width of the top haunch plate chamfer.
2	Height of the top haunch plate chamfer.
3	Height of the bottom haunch plate chamfer.
4	Width of the bottom haunch plate chamfer.

Haunch plate creation

Option	Description
	Default Top and bottom haunch plates are created, if needed. AutoDefaults can change this option.
	Automatic Top or bottom haunch plate or both are created, if needed.
	Top and bottom haunch plates are created. To create a single plate, enter `0` in the thickness (**t**) field for the plate you do not need (top or bottom plate).

Option	Description
	Haunch plates are not created.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

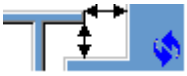
Automatic notching options affect both the top and the bottom flange.

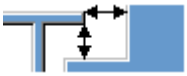
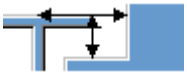
Notch shape

Automatic notching is switched on when you select a notch shape.

Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

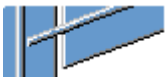
Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.

Option	Description
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

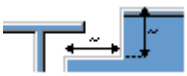
Enter the horizontal and vertical values for the cuts.

Flange cut shape

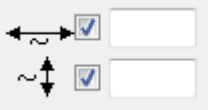
Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.



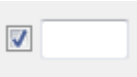
Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.



Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

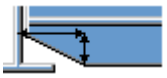
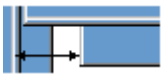
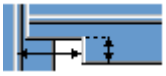
Side of flange notch

The side of flange notch defines on which side of the beam the notches are created.

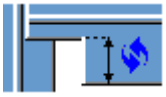
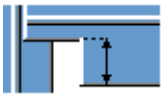
Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

Flange notch shape

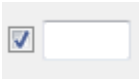
The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

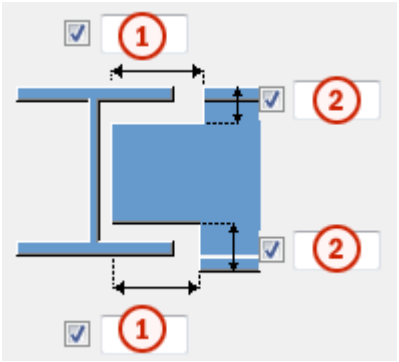
Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.



Cut dimensions



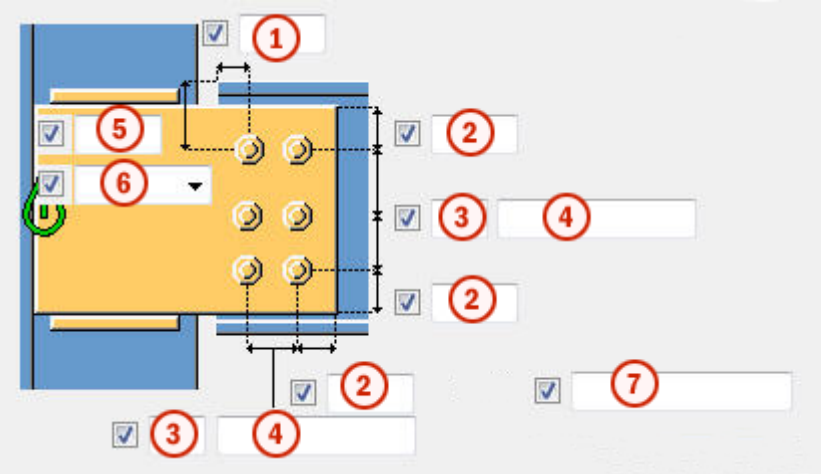
	Description	Default
1	Dimensions for the horizontal flange cuts.	10 mm
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest 5 mm.

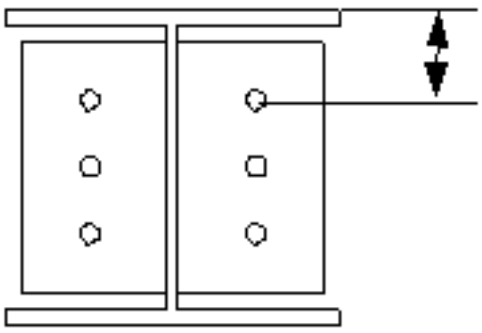
Bolts tab

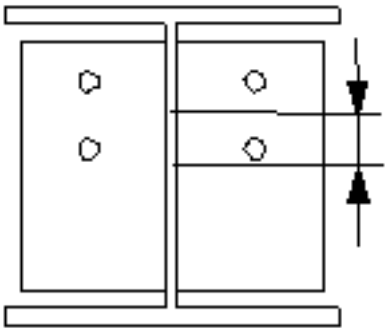
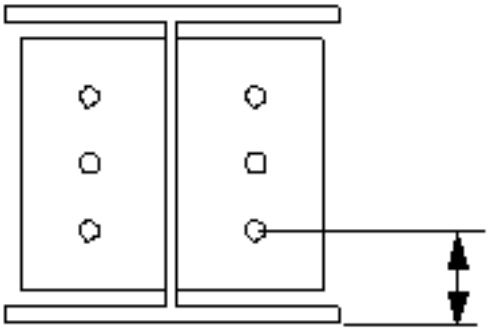
Use the **Bolts** tab to control the properties of the bolts that connect the shear tab to the secondary part.

Bolt group dimensions

Bolt group dimensions affect the size and shape of the shear tab.

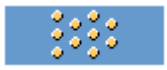


	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position.
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1

Option	Description
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt group orientation

Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Staggered Bolts are staggered in the direction of the secondary part.
	Square Square bolt group is positioned horizontally.
	Sloped Square bolt group is sloped in the direction of the secondary part.

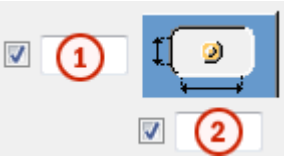
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

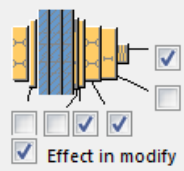


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Beam cut tab

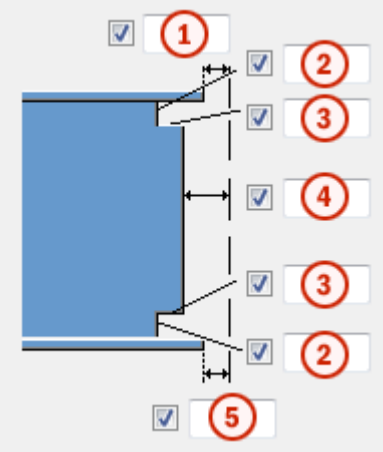
Use the **Beam cut** tab to control weld backing bars, weld access holes, beam end preparations, and flange cuts.

Weld backing bar

Option	Description
Weld backing bar	Weld backing bar thickness and width.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

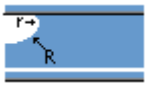
Weld access hole dimensions



	Description
1	Gap between the secondary part top flange and the main part.
2	Vertical dimensions for the top and the bottom weld access holes.
3	Horizontal dimensions for the top and the bottom weld access holes.
4	Gap between the secondary part web and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.
5	Gap between the secondary part bottom flange and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.

Weld access holes

Option	Description	Default
	Default Round weld access hole AutoDefaults can change this option.	
	Round weld access hole	
	Square weld access hole	
	Diagonal weld access hole	

Option	Description	Default
	Round weld access hole with a radius that you can define in r ☑	
	Extended cone-shaped weld access hole with a radius and dimensions that you can define in R ☑ and Top Prep x ☑ Bottom Prep x ☑	
	Cone-shaped weld access hole with radiuses that you can define in R ☑ and r ☑ Capital **R** defines the large radius (height). Small **r** defines the small radius.	R = 35 r = 10

Beam end preparation

Option	Description
	Default Top and bottom flange are prepared. AutoDefaults can change this option.
	Automatic Top and bottom flange are prepared.
	Beam end is not prepared.
	Top and bottom flange are prepared.

Flange cut

Option for top flange	Option for bottom flange	Description
		Default Flange is not cut. AutoDefaults can change this option.
		Flange is not cut.
		Flange is cut.

Weld backing bars

Option for top backing bar	Option for bottom backing bar	Description
		Default Backing bars are created inside the flanges. AutoDefaults can change this option.
		No backing bars are created.
		Backing bars are created inside the flanges.
		Backing bars are created outside the flanges.

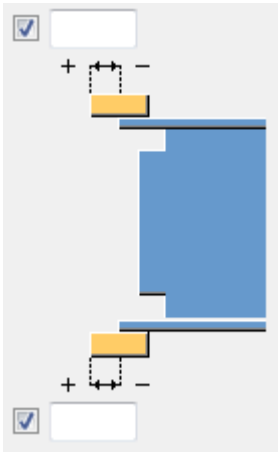
Weld backing bar length

Enter the length of the weld backing bar in the box below the options.

Option	Description
	Default Absolute length of the backing bar AutoDefaults can change this option.
	Absolute length of the backing bar

Option	Description
	Extension beyond the edge of the flange

Weld backing bar position

Option	Description
	Enter a positive or a negative value to move the front end of the backing bar relative to the end of the flange.

Assembly type

Define the location where the weld backing bar welds are made. When you select the **Workshop** option, Tekla Structures includes the backing bars in the assembly.

Doubler plate

Use the **Doubler plate** tab to create doubler plates to strengthen the web of the main part.

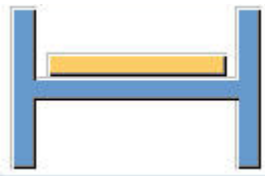
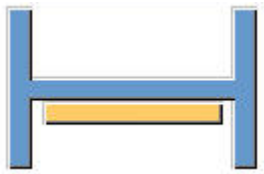
Web plate

Option	Description
Web plate	Web plate thickness and height.

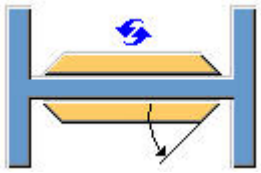
Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in

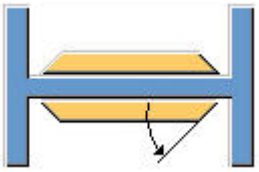
Option	Description	Default
		File menu --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Doubler plates

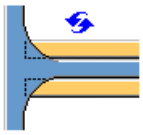
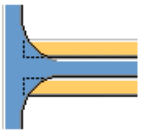
Option	Description
	Default Doubler plates are not created. AutoDefaults can change this option.
	Doubler plates are not created.
	Doubler plate is created on the far side.
	Doubler plate is created on the near side.
	Doubler plates are created on both sides.

Doubler plate edge shape

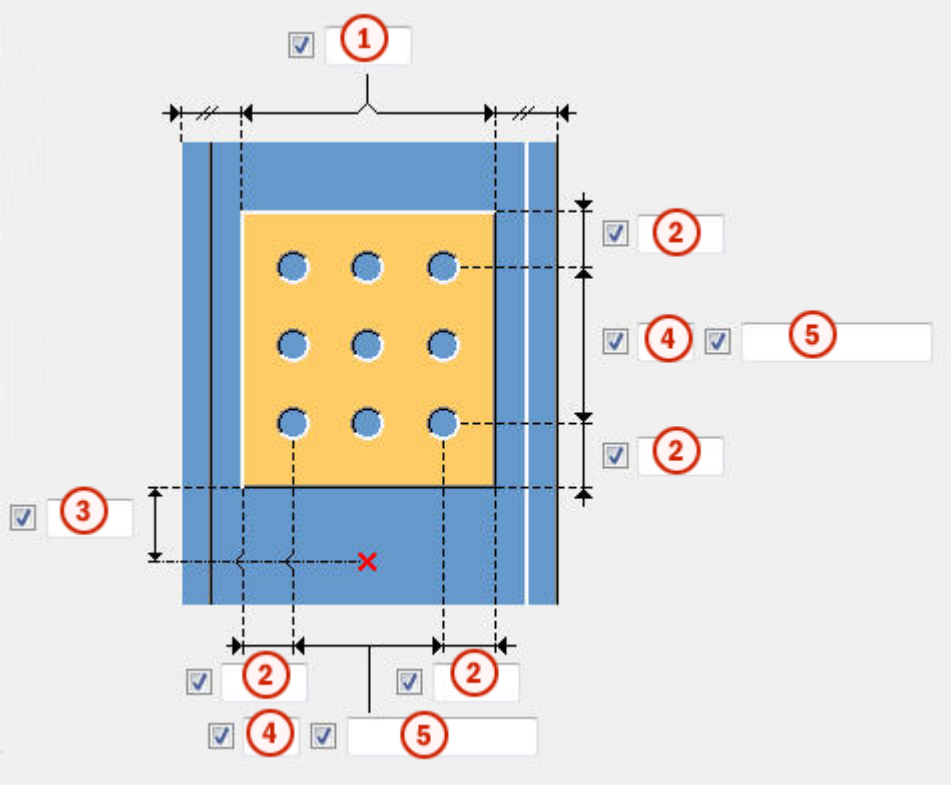
Option	Description
	Default Bevel doubler plates AutoDefaults can change this option.

Option	Description
	Bevel doubler plates Enter the angle in
	Square doubler plates

Doubler plate cuts

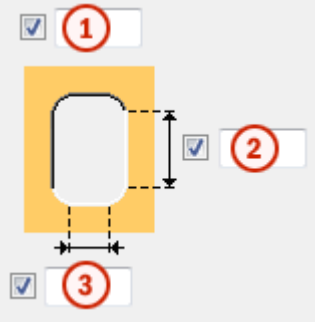
Option	Description
	Default Doubler plates are not cut. AutoDefaults can change this option.
	Doubler plates are not cut.
	Doubler plates are cut in the area that connects the main part web and flange.

General settings



	Description
1	Edge distance from the column flange.
2	Doubler plate edge distance. Edge distance is the distance from the center of a hole to the edge of the part.
3	Edge distance of the doubler plate in relation to the bottom of the secondary part.
4	Number of holes.
5	Hole spacing. Use a space to separate hole spacing values. Enter a value for each space between holes. For example, if there are 3 holes, enter 2 values.

Weld hole size



	Description
1	Hole diameter.
2	Slot length.
3	Slot width.

General tab

Click the link below to find out more:

General tab

Design type tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

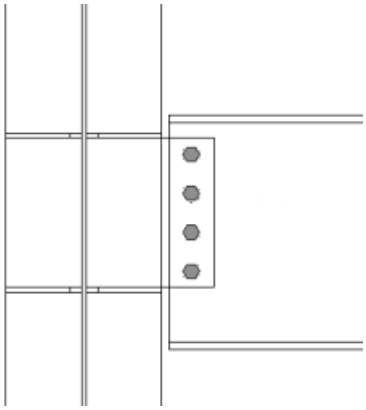
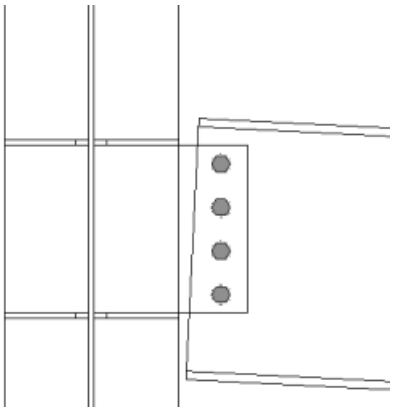
Column with stiffeners (186)

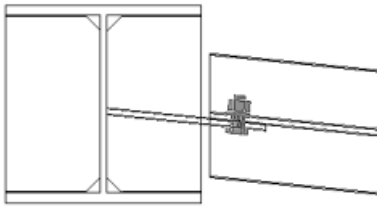
Column with stiffeners (186) connects a beam to a column with a square shear tab. The shear tab is welded to the main part web and stiffeners, and bolted to the secondary part web. The secondary beam can be leveled or sloped.

Objects created

- Shear tabs (1 or 2)
- Stiffeners (optional)
- Haunch plates (optional)
- Web doubler plate (optional)
- Bolts
- Welds
- Cuts

Use for

Situation	Description
	Shear tab with column stiffeners.
	Shear tab with column stiffeners. The secondary part is sloped.

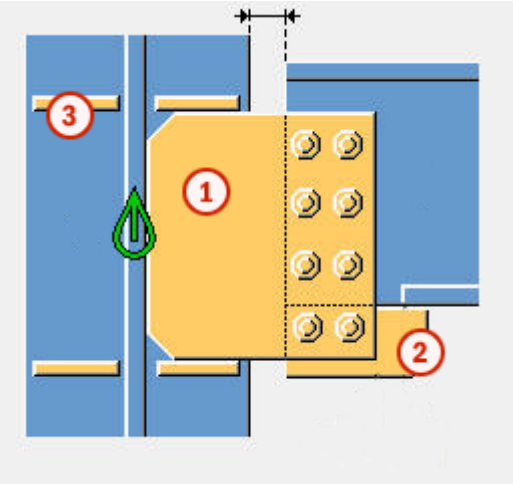
Situation	Description
	Shear tab with column stiffeners. The secondary part is skewed.

Selection order

1. Select the main part (column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key



	Part
1	Shear tab
2	Haunch plate
3	Stiffener

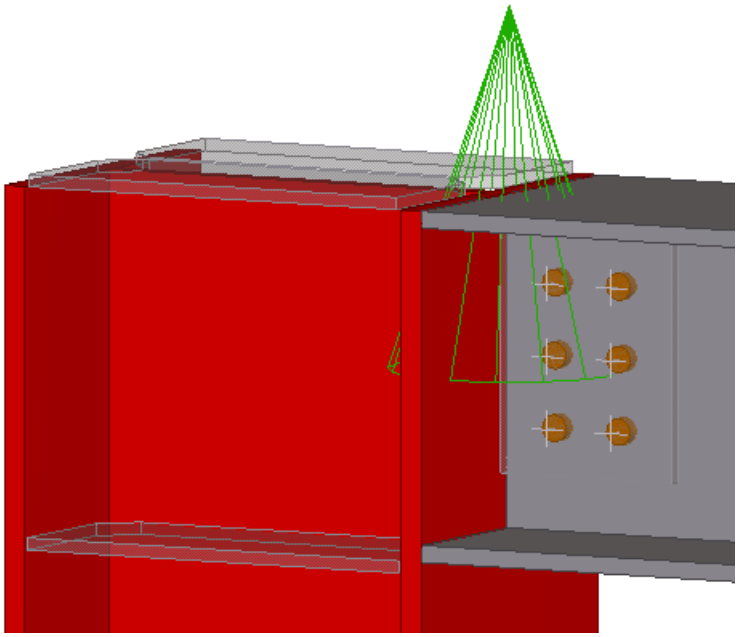
NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Example: Add a beam-to-column connection using the Column with stiffeners (186) connection

In this example, you will connect a beam to a column using a beam-to-column connection.

1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. Enter `186` in the search box.
3. Select **Column with stiffeners (186)**.
4. Select the main part (column).
5. Select the secondary part (beam).

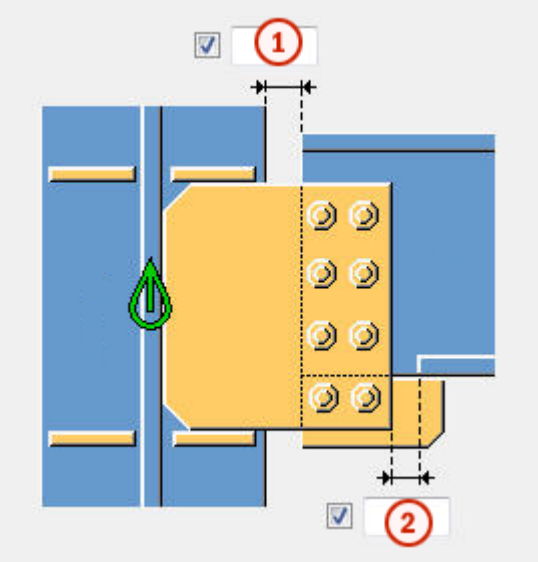
   Tekla Structures automatically adds the connection when you select the beam.



Picture tab

Use the **Picture** tab to control how the secondary beam end is cut.

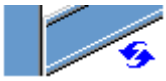
Dimensions



	Description	Default
1	Cut of the secondary part. Cutting the secondary part creates a gap between the main part and the secondary part.	20
2	Size of the strip made to the secondary part flange. The cut of the flange is defined from the shear tab edge.	The flange is automatically stripped when the shear tab crosses the flange. 20

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.

Option	Description
	Square Cuts the end of the secondary beam square.
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.

Beam flange cut

Define how the secondary beam flange end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the flange bevel.
	Square Cuts a part of the flange square and leaves a part of it bevel.

Plates tab

Use the **Plates** tab to control the size, position, number, orientation and shape of the shear tab.

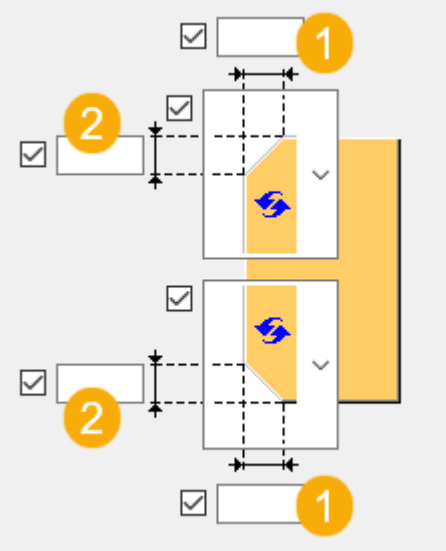
Shear tab plate

Option	Description
Tab plate	Shear tab plate thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

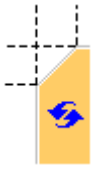
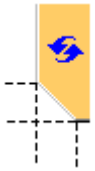
Option	Description	Default
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

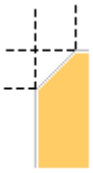
Shear tab chamfers



	Description
1	Horizontal dimension of the shear tab chamfer.
2	Vertical dimension of the shear tab chamfer.

Chamfer type

Option	Option	Description
		Default Line chamfer AutoDefaults can change this option.

Option	Option	Description
		No chamfer
		Line chamfer
		Convex arc chamfer
		Concave arc chamfer

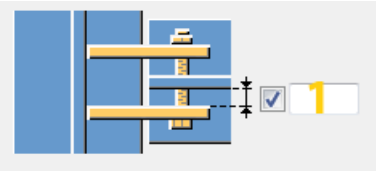
Shear tab position

Define the number and the side of shear tabs in single shear tab connections.

Option	Description
	Default Far side shear tab AutoDefaults can change this option.
	Automatic The component automatically selects either the near side or the far side shear tab. The tab is created to the side of the secondary part when the angle between the main part and the secondary part is less than 90 degrees.
	Far side shear tab

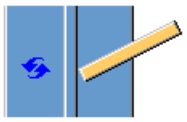
Option	Description
	Near side and far side shear tab
	Near side shear tab

Gap between shear tabs



	Description	Default
1	Gap between the secondary part web and shear tab. This only affects connections with two shear tabs.	0

Shear tab end cut

Option	Description
	Default The shear tab end is not cut. AutoDefaults can change this option.
	Square The shear tab end is not cut.
	Bevel The shear tab end is cut parallel to the main part web.

Shear tab orientation

Option	Description
	Default Sloped AutoDefaults can change this option.

Option	Description
	Automatic The shear tab is sloped in the direction of the secondary beam. Both vertical edges of the shear tab are cut parallel to the end of the secondary beam.
	Sloped The shear tab is sloped in the direction of the secondary beam. Both vertical edges of the shear tab are cut parallel to the end of the secondary beam.
	Square
	Modified sloped Same as the **Sloped** option, but the vertical edge of the shear tab connected to the secondary beam is cut perpendicular to the secondary beam flange.

Stiffeners tab

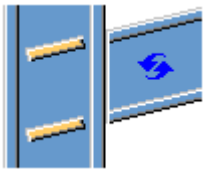
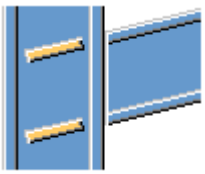
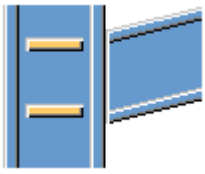
Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position and type.

Stiffener plate dimensions

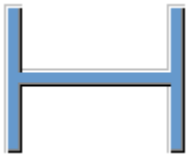
Option	Description
Top NS	Top near side stiffener thickness, width and height.
Top FS	Top far side stiffener thickness, width and height.
Bottom NS	Bottom near side stiffener thickness, width and height.
Bottom FS	Bottom far side stiffener thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Stiffener orientation

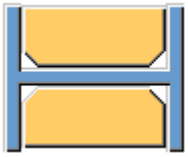
Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are parallel to the secondary part.
	Stiffeners are perpendicular to the main part.

Stiffener creation

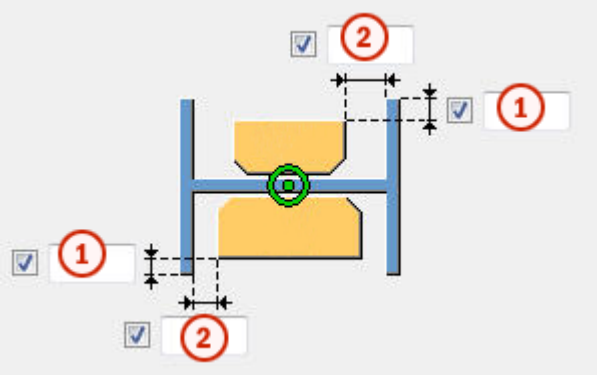
Option	Description
	Default Stiffeners are created. AutoDefaults can change this option.
	Automatic Stiffeners are created when necessary.
	No stiffeners are created.
	Stiffeners are created.

Stiffener shape

Option	Description
	Default Line chamfered stiffener plates AutoDefaults can change this option.
	Automatic Line chamfered stiffener plates
	Square stiffener plates Stiffener plates with a gap for the main part web rounding

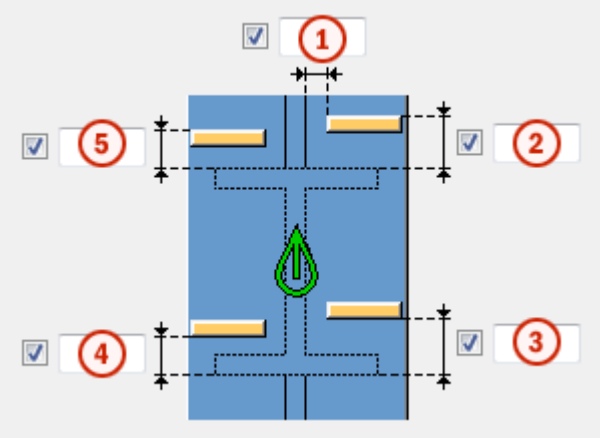
Option	Description
	Line chamfered stiffener plates

Stiffener gap



	Description
1	Distance from the edge of the flange to the edge of the stiffener.
2	Size of the gap between the flanges and the stiffener.

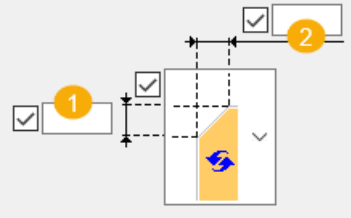
Stiffener positions



	Description
1	Size of the gap between the stiffener and the beam web edge.
2	Size of the gap between the top near side stiffener and the beam flange edge.
3	Size of the gap between the bottom near side stiffener and the beam flange edge.

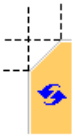
	Description
4	Size of the gap between the bottom far side stiffener and the beam flange edge.
5	Size of the gap between the top far side stiffener and the beam flange edge.

Chamfer dimensions



	Description	Default
1	Vertical dimension of the chamfer.	`10 mm`
2	Horizontal dimension of the chamfer.	`10 mm`

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Option	Description
	Line and arc chamfer

Haunch tab

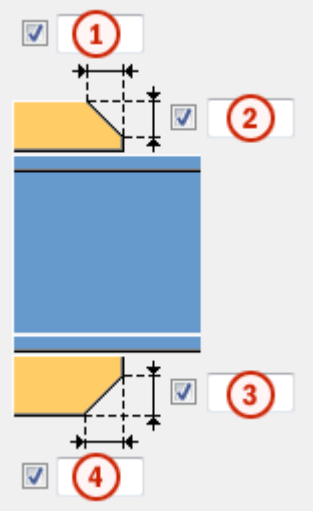
Use the **Haunch** tab to control the haunch plate creation and chamfers in the secondary beam flanges.

Haunch plates

Option	Description
Top plate	Top haunch plate thickness, width and height.
Bottom plate	Bottom haunch plate thickness, width and height.

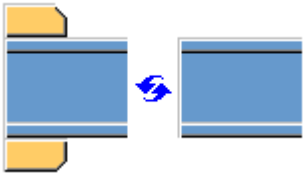
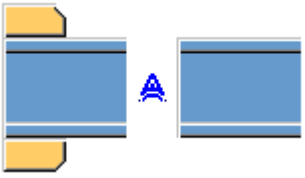
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Haunch plate chamfers



	Description
1	Width of the top haunch plate chamfer.
2	Height of the top haunch plate chamfer.
3	Height of the bottom haunch plate chamfer.
4	Width of the bottom haunch plate chamfer.

Haunch plate creation

Option	Description
	Default Top and bottom haunch plates are created, if needed. AutoDefaults can change this option.
	Automatic Top or bottom haunch plate or both are created, if needed.
	Top and bottom haunch plates are created. To create a single plate, enter `0` in the thickness (**t**) field for the plate you do not need (top or bottom plate).

Option	Description
	Haunch plates are not created.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

Automatic notching

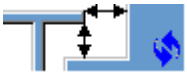
Automatic notching options affect both the top and the bottom flange.

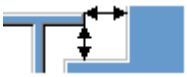
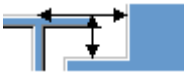
Notch shape

Automatic notching is switched on when you select a notch shape.

Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.
	Turns off automatic notching.

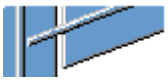
Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.

Option	Description
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

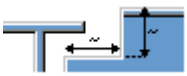
Enter the horizontal and vertical values for the cuts.

Flange cut shape

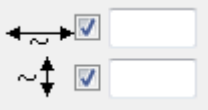
Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.



Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.

Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

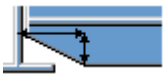
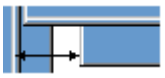
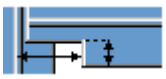
Side of flange notch

The side of flange notch defines on which side of the beam the notches are created.

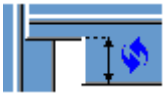
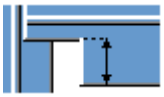
Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

Flange notch shape

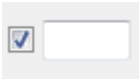
The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

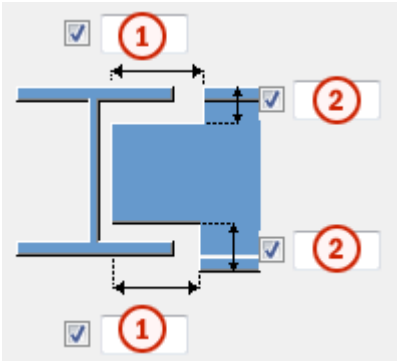
Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.



Cut dimensions



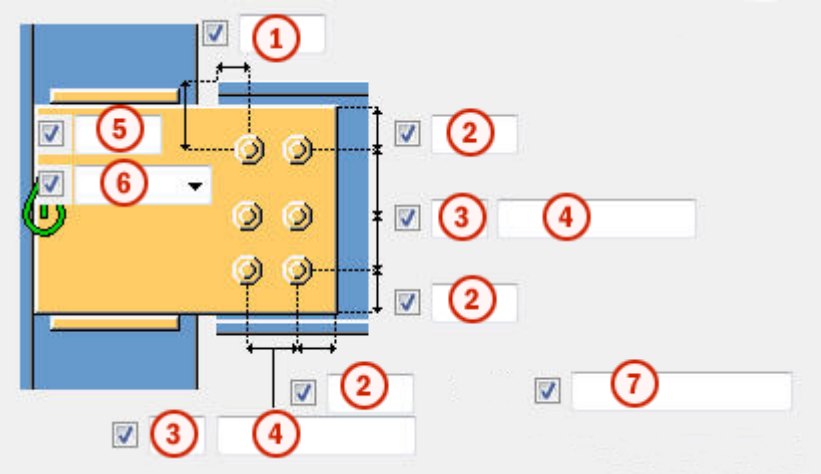
	Description	Default
1	Dimensions for the horizontal flange cuts.	`10 mm`
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest `5 mm`.

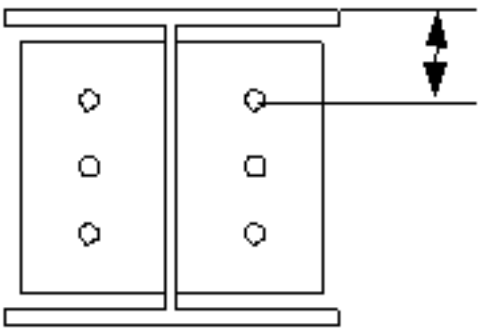
Bolts tab

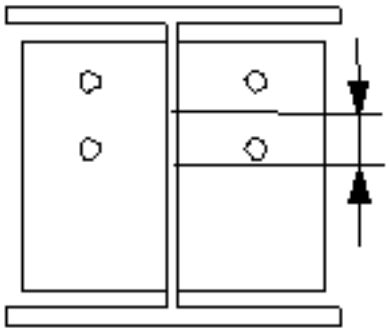
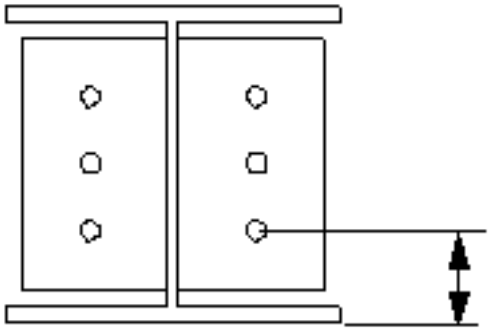
Use the **Bolts** tab to control the properties of the bolts that connect the shear tab to the secondary part.

Bolt group dimensions

Bolt group dimensions affect the size and shape of the shear tab.

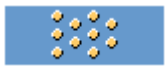


	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position.
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1

Option	Description
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt group orientation

Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Staggered Bolts are staggered in the direction of the secondary part.
	Square Square bolt group is positioned horizontally.
	Sloped Square bolt group is sloped in the direction of the secondary part.

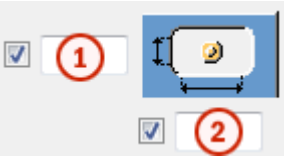
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

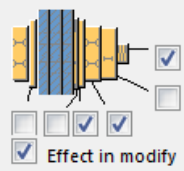


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Doubler plate tab

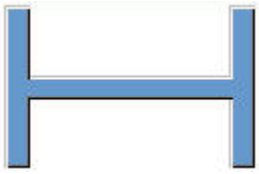
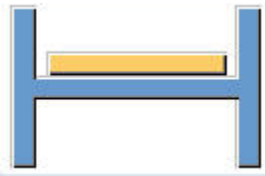
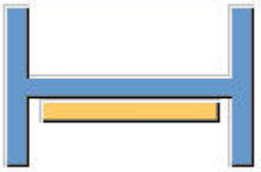
Use the **Doubler plate** tab to create doubler plates to strengthen the web of the main part.

Web plate

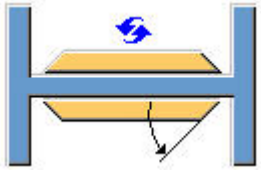
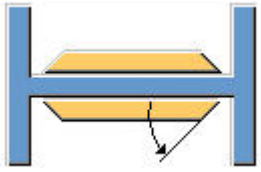
Option	Description
Web plate	Web plate thickness and height.

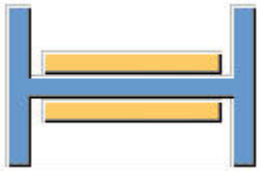
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Doubler plates

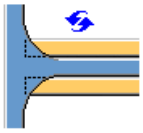
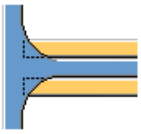
Option	Description
	Default Doubler plates are not created. AutoDefaults can change this option.
	Doubler plates are not created.
	Doubler plate is created on the far side.
	Doubler plate is created on the near side.
	Doubler plates are created on both sides.

Doubler plate edge shape

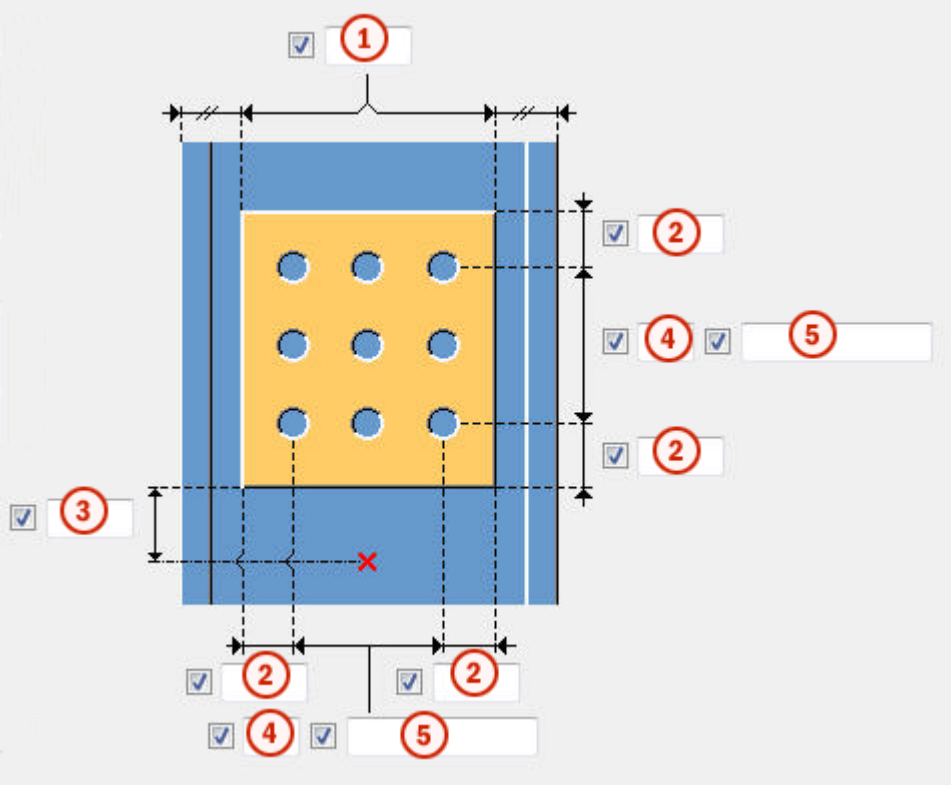
Option	Description
	Default Bevel doubler plates AutoDefaults can change this option.
	Bevel doubler plates Enter the angle in

Option	Description
	Square doubler plates

Doubler plate cuts

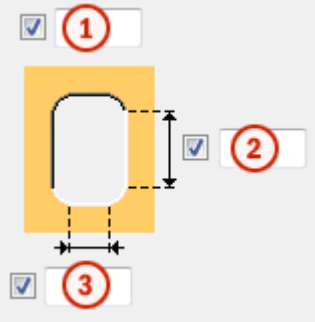
Option	Description
	Default Doubler plates are not cut. AutoDefaults can change this option.
	Doubler plates are not cut.
	Doubler plates are cut in the area that connects the main part web and flange.

General settings



	Description
1	Edge distance from the column flange.
2	Doubler plate edge distance. Edge distance is the distance from the center of a hole to the edge of the part.
3	Edge distance of the doubler plate in relation to the bottom of the secondary part.
4	Number of holes.
5	Hole spacing. Use a space to separate hole spacing values. Enter a value for each space between holes. For example, if there are 3 holes, enter 2 values.

Weld hole size



	Description
1	Hole diameter.
2	Slot length.
3	Slot width.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

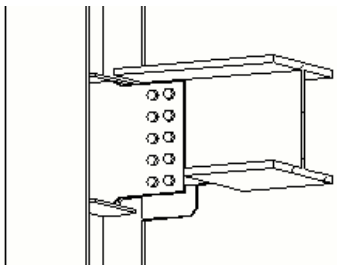
Column with stiffeners S (187)

Column with stiffeners S (187) connects a column to a beam with a shaped shear tab. The shear tab is welded to the main part web and stiffeners, and bolted to the secondary part web. The secondary beam can be leveled or sloped.

Objects created

- Shear tabs (1 or 2)
- Stiffeners (optional)
- Haunch plates (optional)
- Web doubler plate (optional)
- Bolts
- Welds
- Cuts

Use for

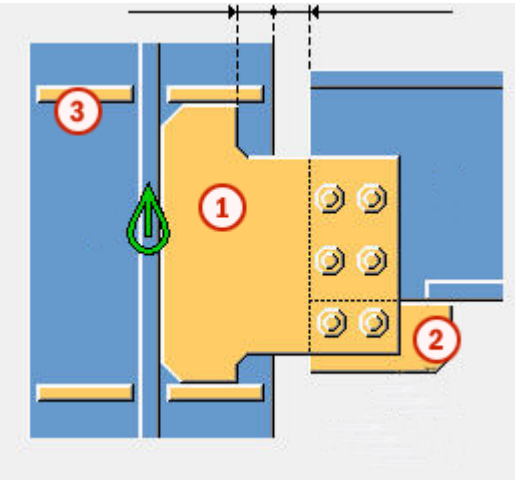
Situation	Description
	Beam connected to column web. Shear tab with bottom haunch plate and four stiffeners.

Selection order

1. Select the main part (column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key



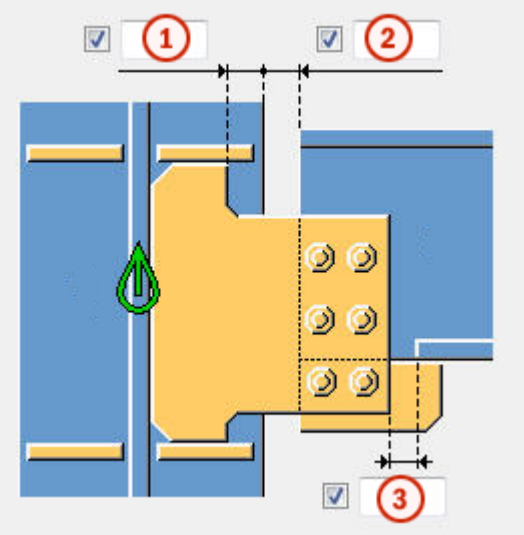
	Part
1	Shear tab
2	Haunch plate
3	Stiffener

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to control the position of the shear tab corner and to define how the beam end is cut.

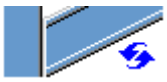
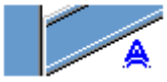
Dimensions



	Description	Default
1	Shear tab edge distance from the main part flange edge.	0
2	Cut of the secondary part. Cutting the secondary part creates a gap between the main part and the secondary part.	20 mm
3	Size of the strip made to the secondary part flange. The cut of the flange is defined from the shear tab edge.	The flange is automatically stripped when the shear tab crosses the flange. 20 mm

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.
	Square Cuts the end of the secondary beam square.
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.

Beam flange cut

Define how the secondary beam flange end is cut. The beam is viewed from the top.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Bevel Cuts the end of the flange bevel.
	Square Cuts a part of the flange square and leaves a part of it bevel.

Plates tab

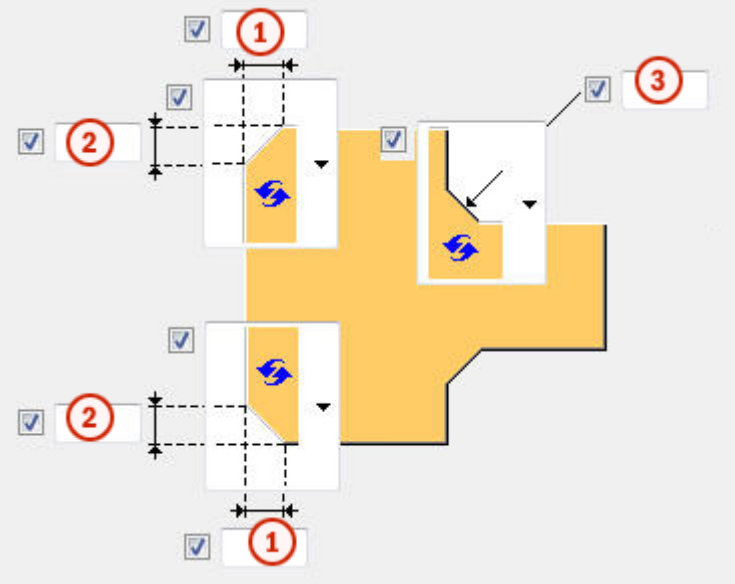
Use the **Plates** tab to control the size, position, number, orientation and shape of the shear tab.

Shear tab plate

Option	Description
Tab plate	Shear tab plate thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

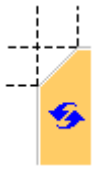
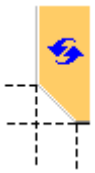
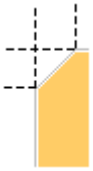
Shear tab chamfers



	Description
1	Horizontal dimension of the shear tab chamfer.
2	Vertical dimension of the shear tab chamfer.

	Description
3	Vertical and the horizontal dimension of the shear tab chamfer.

Chamfer type

Option	Option	Description
		Default Line chamfer AutoDefaults can change this option.
		No chamfer
		Line chamfer
		Convex arc chamfer
		Concave arc chamfer

Chamfer type dimensions

Option	Description
	Default Line chamfer AutoDefaults can change this option.

Option	Description
	No chamfer
	Line chamfer
	Concave arc chamfer

Shear tab orientation

Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Sloped The shear tab is sloped in the direction of the secondary beam. Both vertical edges of the shear tab are cut parallel to the end of the secondary beam.
	Square

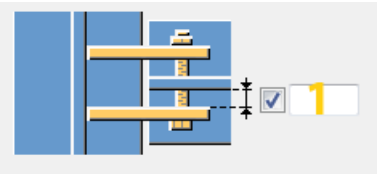
Shear tab position

Define the number and the side of shear tabs in single shear tab connections.

Option	Description
	Default Far side shear tab AutoDefaults can change this option.

Option	Description
	Automatic The component automatically selects either the near side or the far side shear tab. The tab is created to the side of the secondary part when the angle between the main part and the secondary part is less than 90 degrees.
	Far side shear tab
	Near side and far side shear tab
	Near side shear tab

Gap between shear tabs



	Description	Default
1	Gap between the secondary part web and shear tab. This only affects connections with two shear tabs.	0

Stiffeners tab

Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position and type.

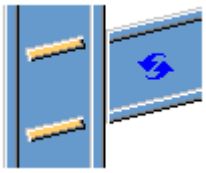
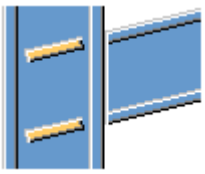
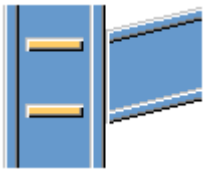
Stiffener plate dimensions

Option	Description
Top NS	Top near side stiffener thickness, width and height.

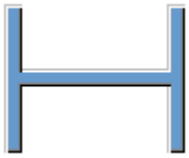
Option	Description
Top FS	Top far side stiffener thickness, width and height.
Bottom NS	Bottom near side stiffener thickness, width and height.
Bottom FS	Bottom far side stiffener thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Stiffener orientation

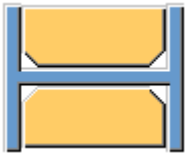
Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are parallel to the secondary part.
	Stiffeners are perpendicular to the main part.

Stiffener creation

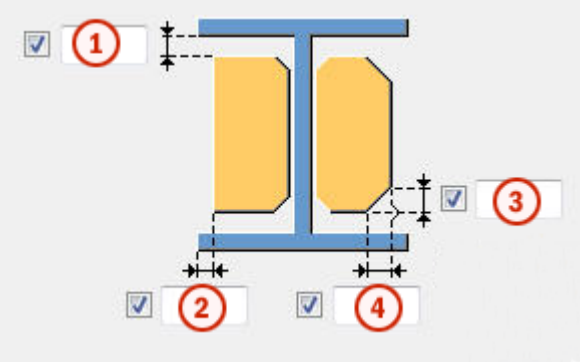
Option	Description
	Default Stiffeners are created. AutoDefaults can change this option.
	Automatic Stiffeners are created when necessary.
	No stiffeners are created.
	Stiffeners are created.

Stiffener shape

Option	Description
	Default Line chamfered stiffener plates AutoDefaults can change this option.
	Automatic Line chamfered stiffener plates
	Square stiffener plates Stiffener plates with a gap for the main part web rounding

Option	Description
	Line chamfered stiffener plates

Stiffener gap and chamfer size



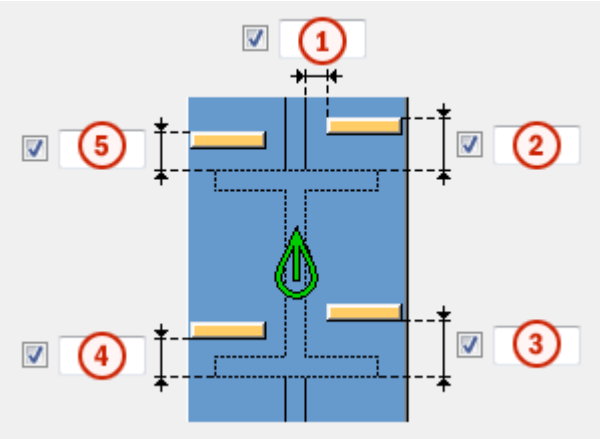
	Description
1	Size of the gap between the flanges and the stiffener.
2	Distance from the edge of the flange to the edge of the stiffener.
3	Vertical dimension of the stiffener line chamfer.
4	Horizontal dimension of the stiffener chamfer or radius of arc type chamfer.

Define the chamfer shape.

Option	Description
	Default No chamfers AutoDefaults can change this option.
	No chamfers
	Line chamfer
	Convex arc chamfer

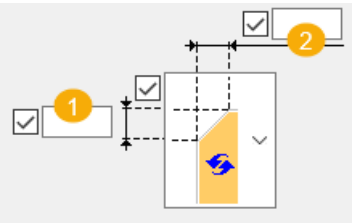
Option	Description
	Concave arc chamfer

Stiffener positions



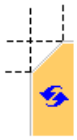
	Description
1	Size of the gap between the stiffener and the beam web edge.
2	Size of the gap between the top near side stiffener and the beam flange edge.
3	Size of the gap between the bottom near side stiffener and the beam flange edge.
4	Size of the gap between the bottom far side stiffener and the beam flange edge.
5	Size of the gap between the top far side stiffener and the beam flange edge.

Chamfer dimensions



	Description	Default
1	Vertical dimension of the chamfer.	`10 mm`
2	Horizontal dimension of the chamfer.	`10 mm`

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer
	Line and arc chamfer

Haunch tab

Use the **Haunch** tab to control the haunch plate creation and chamfers in the secondary beam flanges.

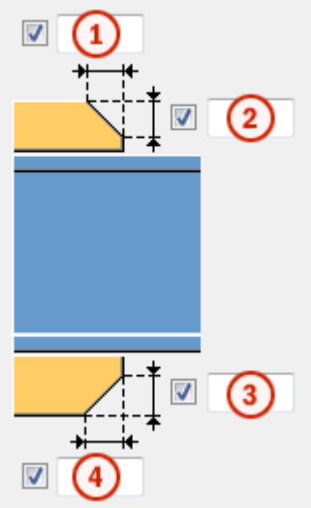
Haunch plates

Option	Description
Top plate	Top haunch plate thickness, width and height.
Bottom plate	Bottom haunch plate thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components**

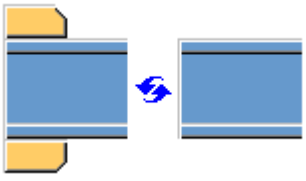
Option	Description	Default
		settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

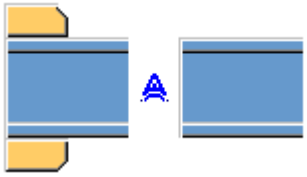
Haunch plate chamfers



	Description
1	Width of the top haunch plate chamfer.
2	Height of the top haunch plate chamfer.
3	Height of the bottom haunch plate chamfer.
4	Width of the bottom haunch plate chamfer.

Haunch plate creation

Option	Description
	Default Top and bottom haunch plates are created, if needed. AutoDefaults can change this option.

Option	Description
	Automatic Top or bottom haunch plate or both are created, if needed.
	Top and bottom haunch plates are created. To create a single plate, enter `0` in the thickness (**t**) field for the plate you do not need (top or bottom plate).
	Haunch plates are not created.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

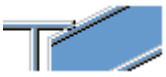
Automatic notching

Automatic notching options affect both the top and the bottom flange.

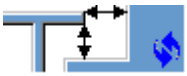
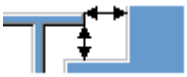
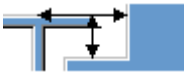
Notch shape

Automatic notching is switched on when you select a notch shape.

Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.

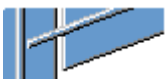
Option	Description
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

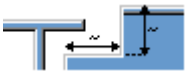
Enter the horizontal and vertical values for the cuts.

Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.

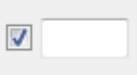
Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.



Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

Side of flange notch

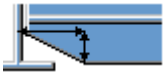
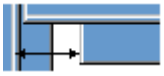
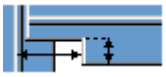
The side of flange notch defines on which side of the beam the notches are created.

Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

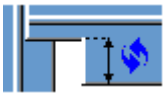
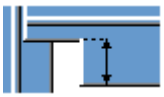
Flange notch shape

The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.

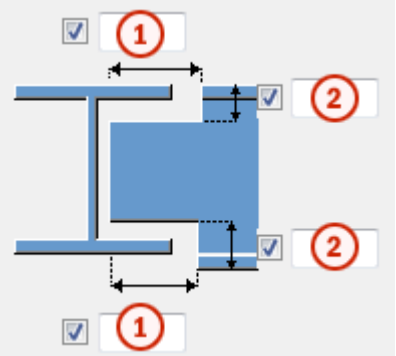
Option	Description
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.

Cut dimensions



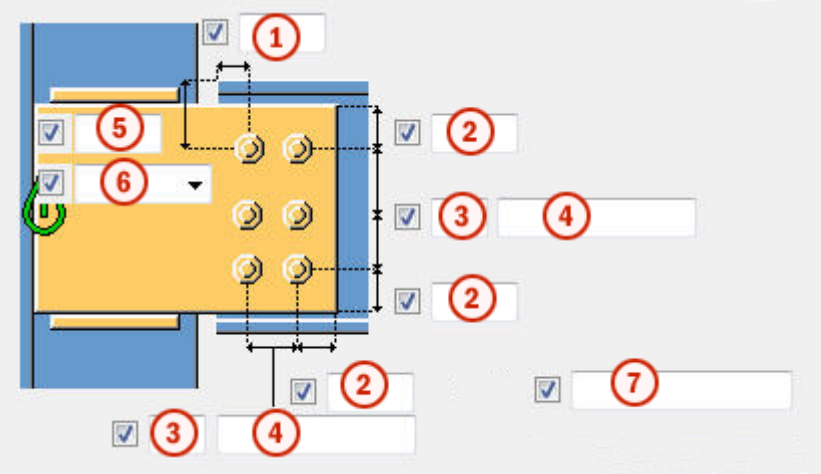
	Description	Default
1	Dimensions for the horizontal flange cuts.	10 mm
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest 5 mm.

Bolts tab

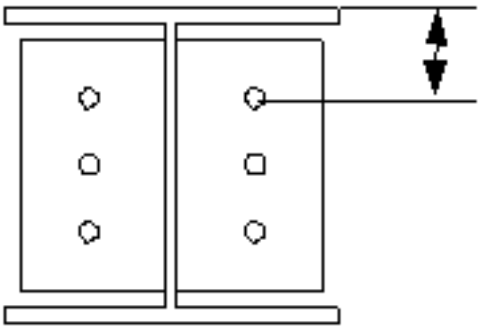
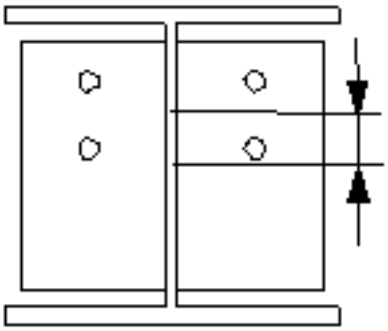
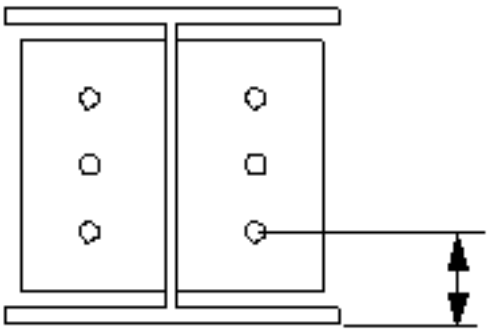
Use the **Bolts** tab to control the properties of the bolts that connect the shear tab to the secondary part.

Bolt group dimensions

Bolt group dimensions affect the size and shape of the shear tab.



	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position.

	Description
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

	Description
7	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt group orientation

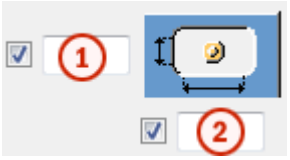
Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Staggered Bolts are staggered in the direction of the secondary part.
	Square Square bolt group is positioned horizontally.
	Sloped Square bolt group is sloped in the direction of the secondary part.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



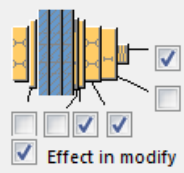
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	

Option	Description	Default
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Doubler plate tab

Use the **Doubler plate** tab to create doubler plates to strengthen the web of the main part.

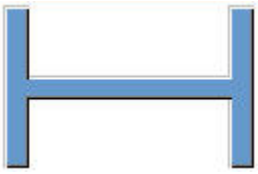
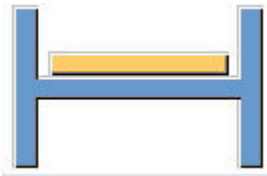
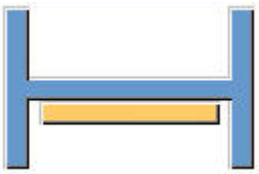
Web plate

Option	Description
Web plate	Web plate thickness and height.

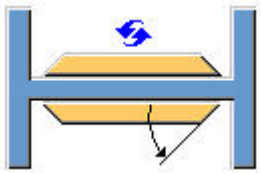
Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the

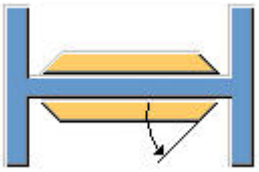
Option	Description	Default
		Components settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Doubler plates

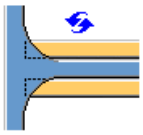
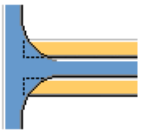
Option	Description
	Default Doubler plates are not created. AutoDefaults can change this option.
	Doubler plates are not created.
	Doubler plate is created on the far side.
	Doubler plate is created on the near side.
	Doubler plates are created on both sides.

Doubler plate edge shape

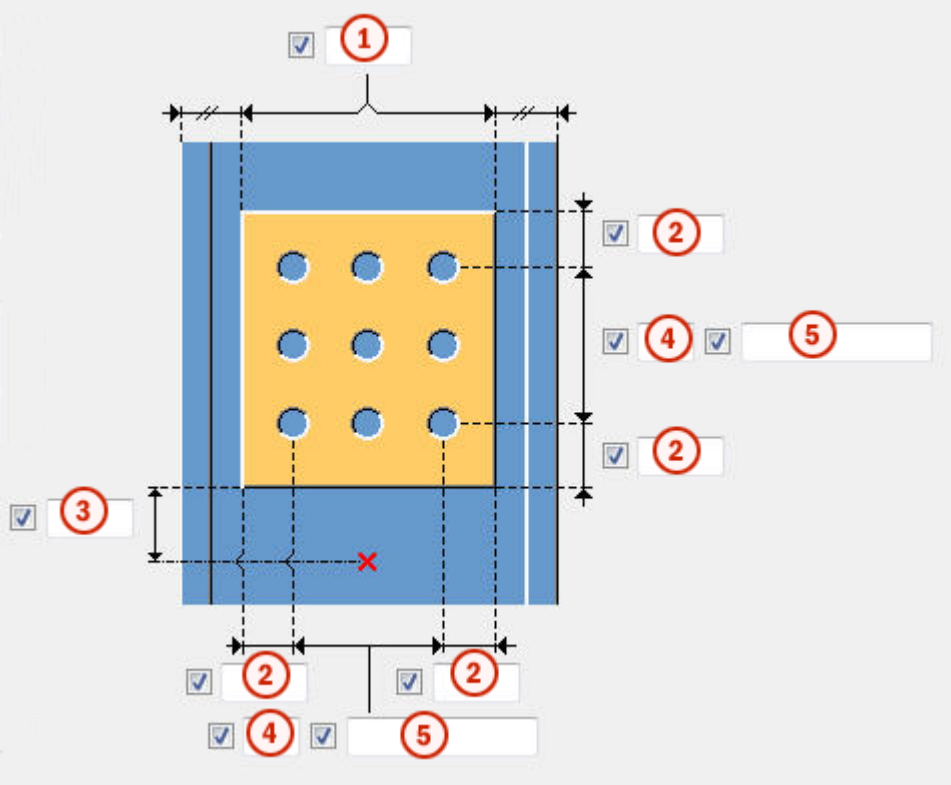
Option	Description
	Default Bevel doubler plates AutoDefaults can change this option.

Option	Description
	Bevel doubler plates Enter the angle in
	Square doubler plates

Doubler plate cuts

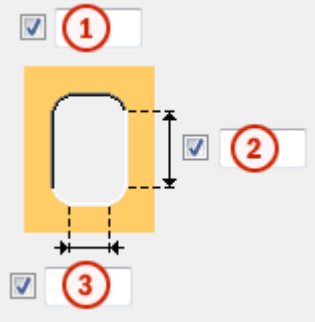
Option	Description
	Default Doubler plates are not cut. AutoDefaults can change this option.
	Doubler plates are not cut.
	Doubler plates are cut in the area that connects the main part web and flange.

General settings



	Description
1	Edge distance from the column flange.
2	Doubler plate edge distance. Edge distance is the distance from the center of a hole to the edge of the part.
3	Edge distance of the doubler plate in relation to the bottom of the secondary part.
4	Number of holes.
5	Hole spacing. Use a space to separate hole spacing values. Enter a value for each space between holes. For example, if there are 3 holes, enter 2 values.

Weld hole size



	Description
1	Hole diameter.
2	Slot length.
3	Slot width.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

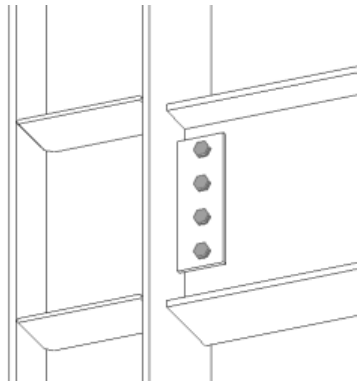
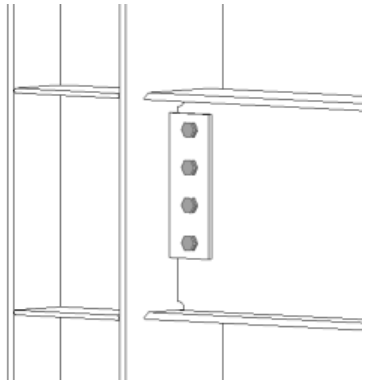
Column with stiffeners (188)

Column with stiffeners (188) connects a column to a beam with a square shear tab. The shear tab is welded to the main part web and stiffeners, and bolted to the secondary part web. The secondary beam can be level or sloped.

Objects created

- Shear tabs (1 or 2)
- Stiffeners (optional)
- Haunch plates (optional)
- Weld backing bars (optional)
- Web doubler plate (optional)
- Bolts
- Welds
- Cuts

Use for

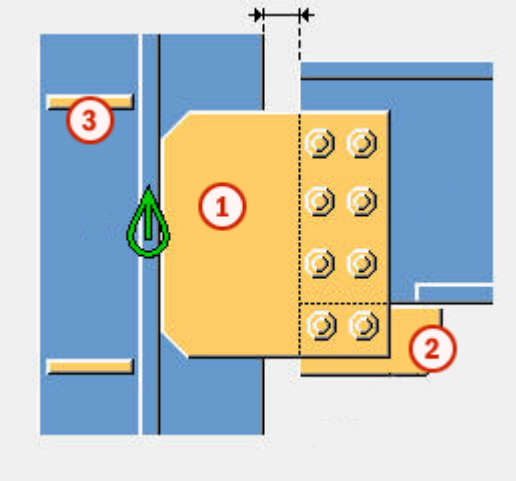
Situation	Description
	Shear tab to column flange with column stiffeners.
	Shear tab to column flange with column stiffeners. Weld preparation and weld access holes for moment connection.

Selection order

1. Select the main part (column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key



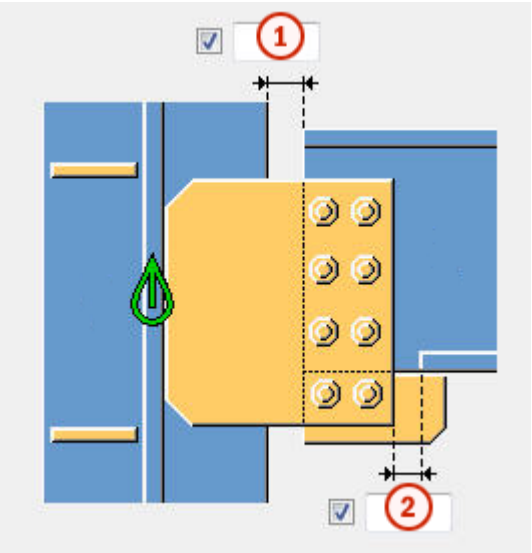
	Part
1	Shear tab
2	Haunch plate
3	Stiffener

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to control how the beam end is cut.

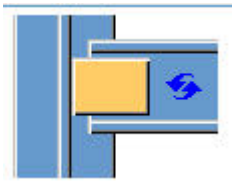
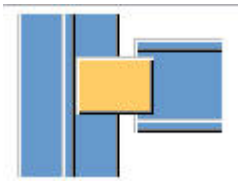
Dimensions



	Description	Default
1	Cut of the secondary part. Cutting the secondary part creates a gap between the main part and the secondary part. The cut is defined from the main part web or the main part flange.	20 mm
2	Size of the strip made to the secondary part flange. The cut of the flange is defined from the shear tab edge.	The flange is automatically stripped when the shear tab crosses the flange. 20 mm

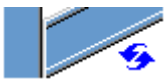
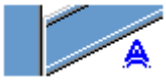
Secondary part cut

Define whether the secondary part cut is calculated from the main part web or main part flange.

Option	Description
	Default From the main part web AutoDefaults can change this option.
	From the main part flange

Beam end cut

Define how the secondary beam end is cut. The beam is viewed from the side.

Option	Description
	Default Bevel AutoDefaults can change this option.
	Automatic If the secondary beam is sloped less than 10 degrees, the beam end is cut square. Otherwise, the beam end is cut bevel.

Option	Description
	Square Cuts the end of the secondary beam square.
	Bevel Cuts the end of the secondary beam parallel to the edge of the main part.

Plates tab

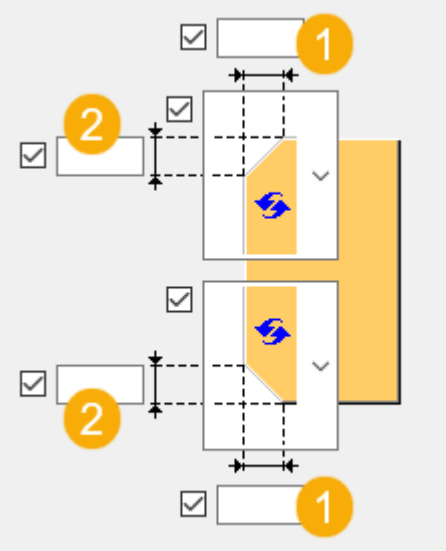
Use the **Plates** tab to control the size, position, number, orientation and shape of the shear tab.

Shear tab plate

Option	Description
Tab plate	Shear tab plate thickness, width and height.

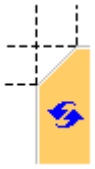
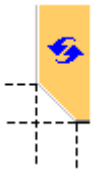
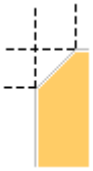
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Shear tab chamfers



	Description
1	Horizontal dimension of the shear tab chamfer.
2	Vertical dimension of the shear tab chamfer.

Chamfer type

Option	Option	Description
		Default Line chamfer AutoDefaults can change this option.
		No chamfer
		Line chamfer
		Convex arc chamfer

Option	Option	Description
		Concave arc chamfer

Shear tab orientation

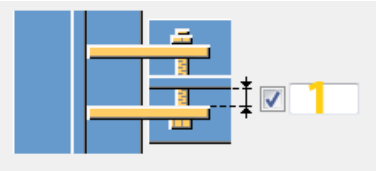
Option	Description
	Default Sloped AutoDefaults can change this option.
	Automatic The shear tab is sloped in the direction of the secondary beam. Both vertical edges of the shear tab are cut parallel to the end of the secondary beam.
	Sloped The shear tab is sloped in the direction of the secondary beam. Both vertical edges of the shear tab are cut parallel to the end of the secondary beam.
	Square
	Modified sloped Same as the **Sloped** option, but the vertical edge of the shear tab connected to the secondary beam is cut perpendicular to the secondary beam flange.

Shear tab position

Define the number and the side of shear tabs in single shear tab connections.

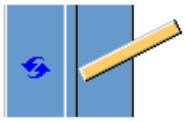
Option	Description
	Default Far side shear tab AutoDefaults can change this option.
	Automatic The component automatically selects either the near side or the far side shear tab. The tab is created to the side of the secondary part when the angle between the main part and the secondary part is less than 90 degrees.
	Far side shear tab
	Near side and far side shear tab
	Near side shear tab

Gap between shear tabs



	Description	Default
1	Gap between the secondary part web and shear tab. This only affects connections with two shear tabs.	0

Shear tab end cut

Option	Description
	Default The shear tab end is not cut. AutoDefaults can change this option.
	Square The shear tab end is not cut.
	Bevel The shear tab end is cut parallel to the main part web.

Stiffeners tab

Use the **Stiffeners** tab to control the stiffener plate dimensions, orientation, position and type.

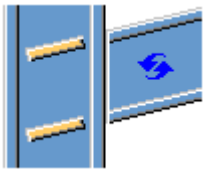
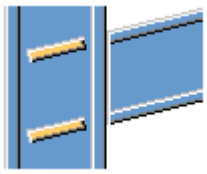
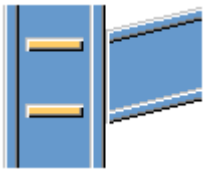
Stiffener plate dimensions

Option	Description
Top NS	Top near side stiffener thickness, width and height.
Top FS	Top far side stiffener thickness, width and height.
Bottom NS	Bottom near side stiffener thickness, width and height.
Bottom FS	Bottom far side stiffener thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
Name	Name that is shown in drawings and reports.	

Stiffener orientation

Option	Description
	Default Stiffeners are parallel to the secondary part. AutoDefaults can change this option.
	Stiffeners are parallel to the secondary part.
	Stiffeners are perpendicular to the main part.

Stiffener creation

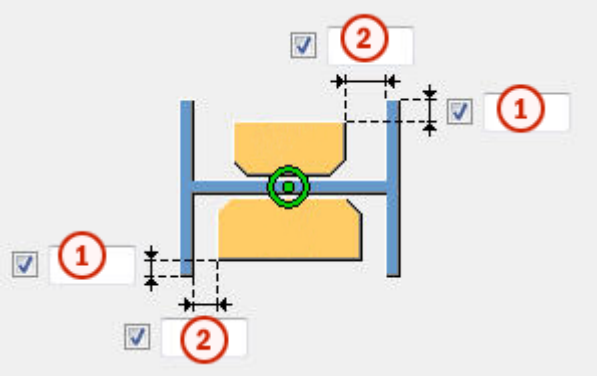
Option	Description
	Default Stiffeners are created. AutoDefaults can change this option.
	Automatic Stiffeners are created when necessary.
	No stiffeners are created.

Option	Description
	Stiffeners are created.

Stiffener shape

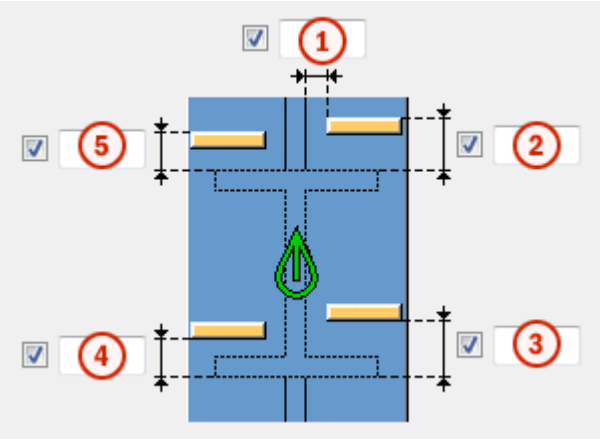
Option	Description
	Default Line chamfered stiffener plates AutoDefaults can change this option.
	Automatic Line chamfered stiffener plates
	Square stiffener plates Stiffener plates with a gap for the main part web rounding
	Line chamfered stiffener plates

Stiffener gap



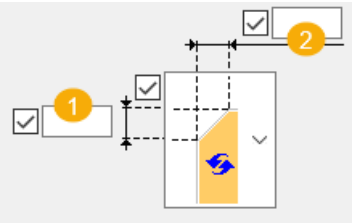
	Description
1	Distance from the edge of the flange to the edge of the stiffener.
2	Size of the gap between the flanges and the stiffener.

Stiffener positions



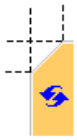
	Description
1	Size of the gap between the stiffener and the beam web edge.
2	Size of the gap between the top near side stiffener and the beam flange edge.
3	Size of the gap between the bottom near side stiffener and the beam flange edge.
4	Size of the gap between the bottom far side stiffener and the beam flange edge.
5	Size of the gap between the top far side stiffener and the beam flange edge.

Chamfer dimensions



	Description	Default
1	Vertical dimension of the chamfer.	`10 mm`
2	Horizontal dimension of the chamfer.	`10 mm`

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer
	Line and arc chamfer

Haunch tab

Use the **Haunch** tab to control the haunch plate creation and chamfers in the secondary beam flanges.

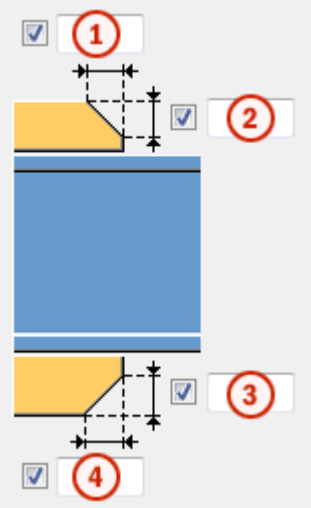
Haunch plates

Option	Description
Top plate	Top haunch plate thickness, width and height.
Bottom plate	Bottom haunch plate thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components**

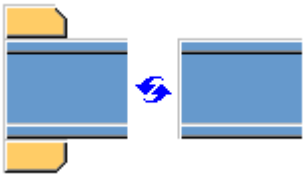
Option	Description	Default
		settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

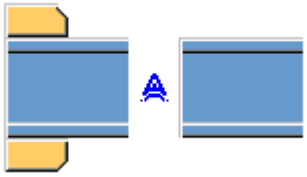
Haunch plate chamfers



	Description
1	Width of the top haunch plate chamfer.
2	Height of the top haunch plate chamfer.
3	Height of the bottom haunch plate chamfer.
4	Width of the bottom haunch plate chamfer.

Haunch plate creation

Option	Description
	Default Top and bottom haunch plates are created, if needed. AutoDefaults can change this option.

Option	Description
	Automatic Top or bottom haunch plate or both are created, if needed.
	Top and bottom haunch plates are created. To create a single plate, enter `0` in the thickness (**t**) field for the plate you do not need (top or bottom plate).
	Haunch plates are not created.

Notch tab

Use the **Notch** tab to automatically create notches for the secondary beam and to control the notch properties. The **Notch** tab has two sections: automatic properties (top section) and manual properties (bottom section). Automatic and manual notching properties work independently from each other.

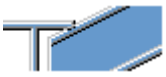
Automatic notching

Automatic notching options affect both the top and the bottom flange.

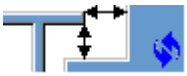
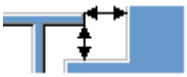
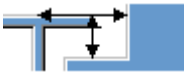
Notch shape

Automatic notching is switched on when you select a notch shape.

Option	Description
	Default Creates notches to the secondary beam. AutoDefaults can change this option.
	Creates notches to the secondary beam. The cuts are square to the main beam web.
	Creates notches to the secondary beam. The cuts are square to the secondary beam web.
	Creates notches to the secondary beam. The vertical cut is square to the main beam, and the horizontal cut is square to the secondary beam.

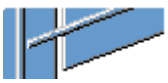
Option	Description
	Turns off automatic notching.

Notch size

Option	Description
	Default The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam. AutoDefaults can change this option.
	The notch size is measured from the edge of the main beam flange and from underneath the top flange of the main beam.
	The notch size is measured from the center line of the main beam and from the top flange of the main beam.

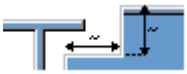
Enter the horizontal and vertical values for the cuts.

Flange cut shape

Option	Description
	Default Secondary beam flange is cut parallel to the main beam. AutoDefaults can change this option.
	Secondary beam flange is cut parallel to the main beam.
	Secondary beam flange is cut square.

Notch dimension rounding

Use the notch dimension rounding options to define whether the notch dimensions are rounded up. Even if the dimension rounding is set to active, the dimensions are rounded up only when necessary.

Option	Description
	Default Notch dimensions are not rounded. AutoDefaults can change this option.
	Notch dimensions are not rounded.
	Notch dimensions are rounded. Enter the horizontal and vertical rounding values.

The dimensions are rounded up the nearest multiple of the value you enter. For example, if the actual dimension is 51 and you enter a round-up value of 10, the dimension is rounded up to 60.

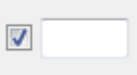
Notch position

Option	Description
	Default Creates the cut below the main beam flange. AutoDefaults can change this option.
	Creates the cut below the main beam flange.
	Creates the cut above the main beam flange.

Notch chamfer

Option	Description
	Default The notch is not chamfered. AutoDefaults can change this option.
	The notch is not chamfered.
	Creates the notch with a line chamfer.
	The notch is chamfered according to the radius you enter.

Enter a radius for the chamfer.



Manual notching

Use manual notching when a part that does not belong to the connection clashes with the secondary beam. When you use manual notching, the connection creates cuts using the values you enter in the fields on the **Notch** tab. You can use different values for the top and the bottom flange.

Side of flange notch

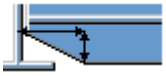
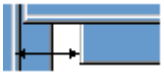
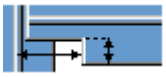
The side of flange notch defines on which side of the beam the notches are created.

Option	Description
	Default Creates notches on both sides of the flange. AutoDefaults can change this option.
	Automatic Creates notches on both sides of the flange.
	Creates notches on both sides of the flange.
	Creates notches on the near side of the flange.
	Creates notches on the far side of the flange.

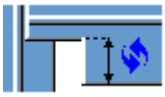
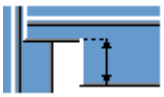
Flange notch shape

The flange notch shape defines the notch shape in the beam flange.

Option	Description
	Default The entire flange of the secondary beam is cut as far back as you define. AutoDefaults can change this option.
	Automatic The entire flange of the secondary beam is cut as far back as you define. The default depth for the notch is twice the thickness of the secondary flange. The cut always runs the entire width of the secondary flange.

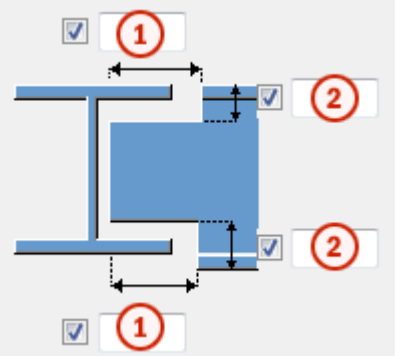
Option	Description
	Creates chamfers in the flange. If you do not enter a horizontal dimension, a chamfer of 45 degrees is created.
	Creates cuts to the flange with default values unless you enter values in the fields **1**and **2**.
	The flange is not cut.
	Creates cuts to the flange according to the value in the field **1** to make it flush with the web.
	Creates cuts to the flange according to the values in the fields **1** and **2**.

Flange notch depth

Option	Description
	Default Flange notch depth. AutoDefaults can change this option.
	Flange notch depth.
	Flange notch depth with a dimension from the secondary beam web center line to the edge of the notch.

Enter the value for flange notch depth.

Cut dimensions



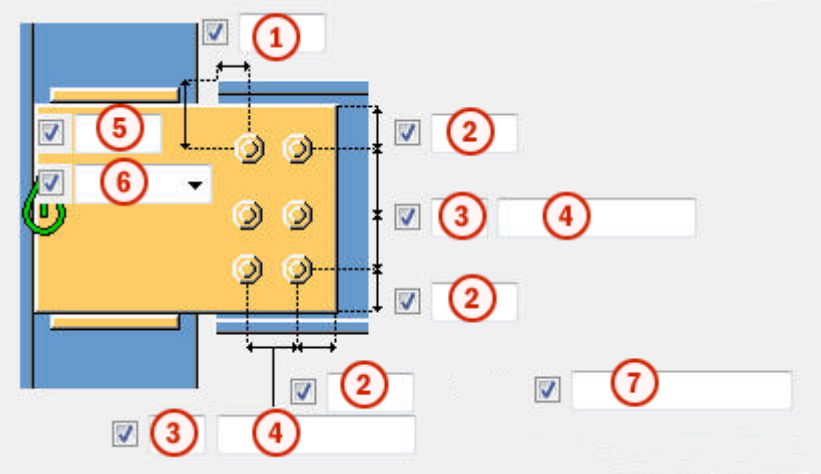
	Description	Default
1	Dimensions for the horizontal flange cuts.	10 mm
2	Dimensions for the vertical flange cuts.	The gap between the notch edge and the beam flange is equal to the main part web rounding. The notch height is rounded up to the nearest 5 mm.

Bolts tab

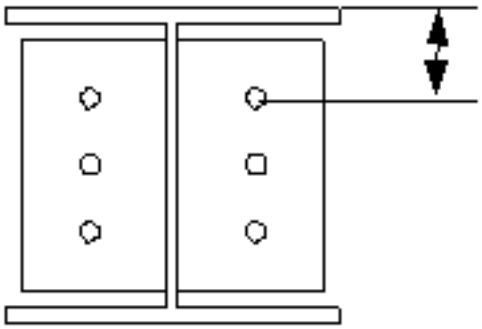
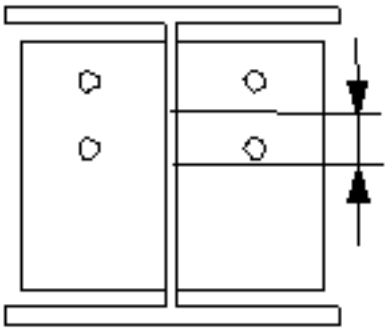
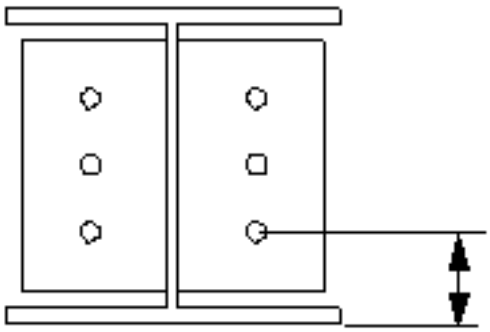
Use the **Bolts** tab to control the properties of the bolts that connect the shear tab to the secondary part.

Bolt group dimensions

Bolt group dimensions affect the size and shape of the shear tab.



	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Dimension for vertical bolt group position.

	Description
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

	Description
7	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt group orientation

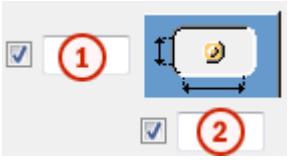
Option	Description
	Default Square AutoDefaults can change this option.
	Automatic Square
	Staggered Bolts are staggered in the direction of the secondary part.
	Square Square bolt group is positioned horizontally.
	Sloped Square bolt group is sloped in the direction of the secondary part.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



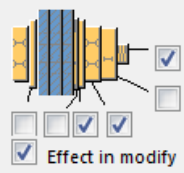
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	

Option	Description	Default
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Beam cut tab

Use the **Beam cut** tab to control weld backing bars, weld access holes, beam end preparations, and flange cuts.

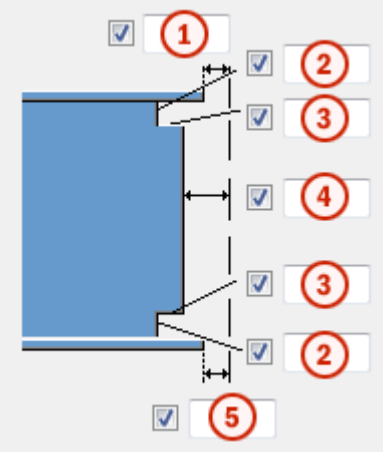
Weld backing bar

Option	Description
Weld backing bar	Weld backing bar thickness and width.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

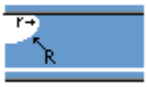
Option	Description	Default
	assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Weld access hole dimensions



	Description
1	Gap between the secondary part top flange and the main part.
2	Vertical dimensions for the top and the bottom weld access holes.
3	Horizontal dimensions for the top and the bottom weld access holes.
4	Gap between the secondary part web and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.
5	Gap between the secondary part bottom flange and the main part. Tekla Structures adds the value you enter here to the gap you enter on the **Picture** tab.

Weld access holes

Option	Description	Default
	Default Round weld access hole AutoDefaults can change this option.	
	Round weld access hole	
	Square weld access hole	
	Diagonal weld access hole	
	Round weld access hole with a radius that you can define in r ☑	
	Extended cone-shaped weld access hole with a radius and dimensions that you can define in R ☑ and Top Prep x ☑ Bottom Prep x ☑	
	Cone-shaped weld access hole with radiuses that you can define in R ☑ and r ☑ Capital **R** defines the large radius (height). Small **r** defines the small radius.	R = 35 r = 10

Beam end preparation

Option	Description
	Default Top and bottom flange are prepared. AutoDefaults can change this option.
	Automatic Top and bottom flange are prepared.
	Beam end is not prepared.
	Top and bottom flange are prepared.

Flange cut

Option for top flange	Option for bottom flange	Description
		Default Flange is not cut. AutoDefaults can change this option.
		Flange is not cut.
		Flange is cut.

Weld backing bars

Option for top backing bar	Option for bottom backing bar	Description
		Default Backing bars are created inside the flanges. AutoDefaults can change this option.
		No backing bars are created.

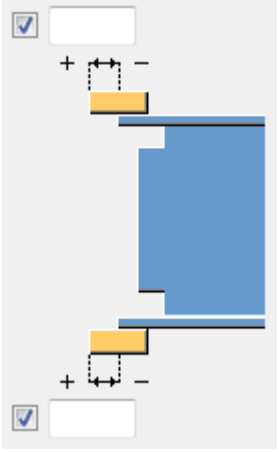
Option for top backing bar	Option for bottom backing bar	Description
		Backing bars are created inside the flanges.
		Backing bars are created outside the flanges.

Weld backing bar length

Enter the length of the weld backing bar in the box below the options.

Option	Description
	Default Absolute length of the backing bar AutoDefaults can change this option.
	Absolute length of the backing bar
	Extension beyond the edge of the flange

Weld backing bar position

Option	Description
	Enter a positive or a negative value to move the front end of the backing bar relative to the end of the flange.

Assembly type

Define the location where the weld backing bar welds are made. When you select the **Workshop** option, Tekla Structures includes the backing bars in the assembly.

Doubler plate tab

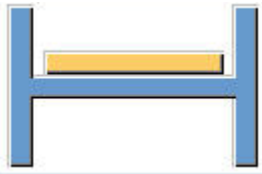
Use the **Doubler plate** tab to create doubler plates to strengthen the web of the main part.

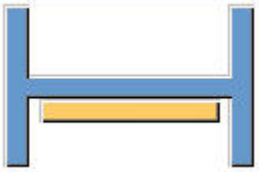
Web plate

Option	Description
Web plate	Web plate thickness and height.

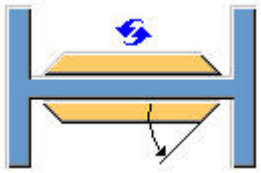
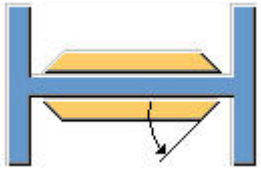
Option	Description	Default
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Doubler plates

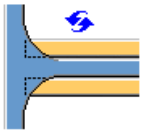
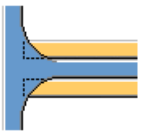
Option	Description
	Default Doubler plates are not created. AutoDefaults can change this option.
	Doubler plates are not created.
	Doubler plate is created on the far side.

Option	Description
	Doubler plate is created on the near side.
	Doubler plates are created on both sides.

Doubler plate edge shape

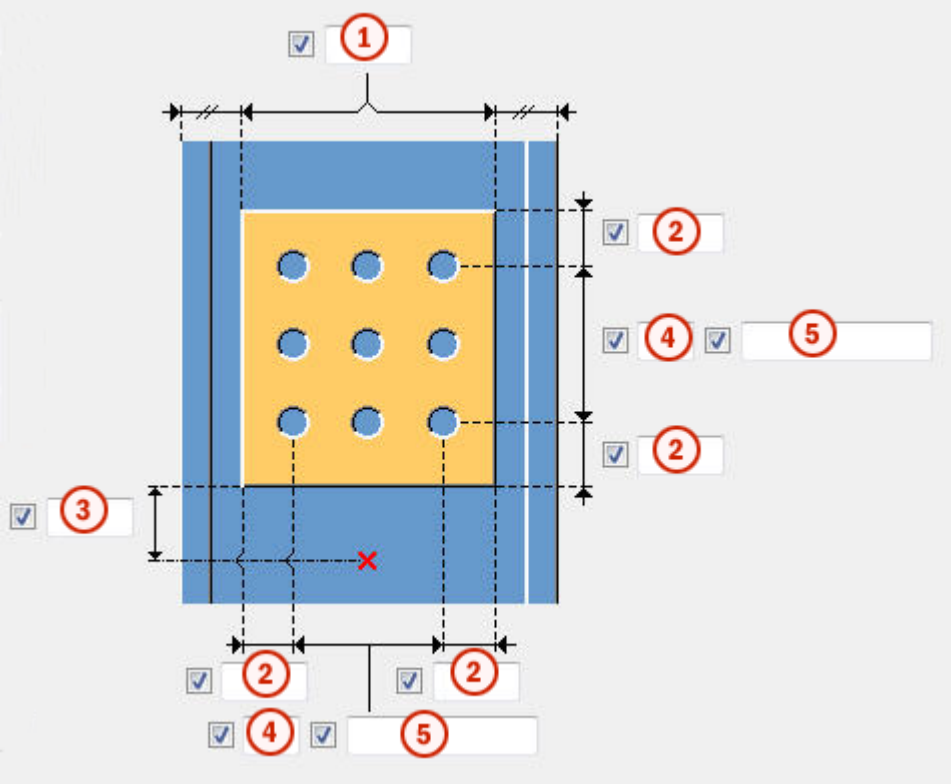
Option	Description
	Default Bevel doubler plates AutoDefaults can change this option.
	Bevel doubler plates Enter the angle in
	Square doubler plates

Doubler plate cuts

Option	Description
	Default Doubler plates are not cut. AutoDefaults can change this option.
	Doubler plates are not cut.

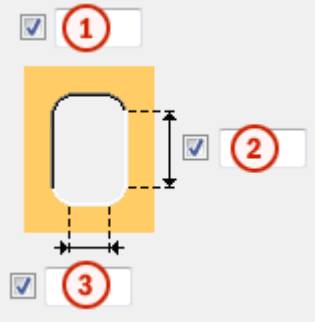
Option	Description
	Doubler plates are cut in the area that connects the main part web and flange.

General settings



	Description
1	Edge distance from the column flange.
2	Doubler plate edge distance. Edge distance is the distance from the center of a hole to the edge of the part.
3	Edge distance of the doubler plate in relation to the bottom of the secondary part.
4	Number of holes.
5	Hole spacing. Use a space to separate hole spacing values. Enter a value for each space between holes. For example, if there are 3 holes, enter 2 values.

Weld hole size



	Description
1	Hole diameter.
2	Slot length.
3	Slot width.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

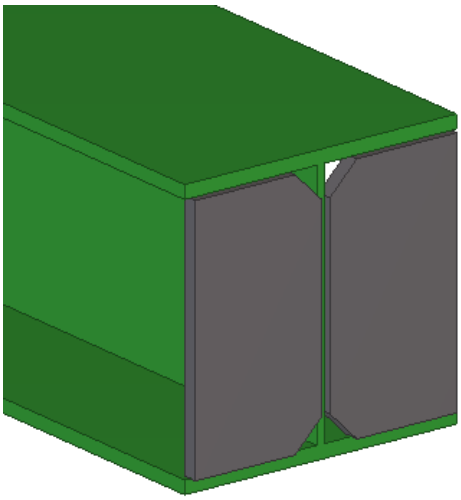
Stiffeners (1003)

Stiffeners (1003) creates stiffeners to I-profiles.

Objects created

- Stiffeners

Use for

Situation	Description
	Stiffeners at a beam end.

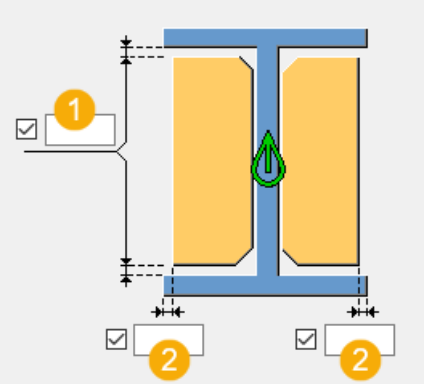
Selection order

1. Select the main part (beam).
2. Pick a position.

   The detail is created automatically when the position is picked.

Picture tab

Use the **Picture** tab to control the stiffener plate position.



	Description
1	Define the gap between the stiffener plates and beam flanges. The default size of the gap between the stiffener plates and beam flanges is `0 mm`.

	Description
2	Define the stiffener plate distance from the beam flange.

Parts tab

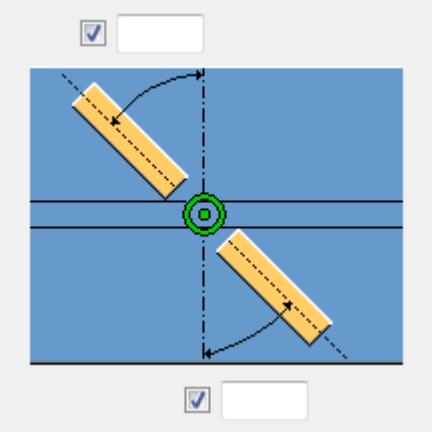
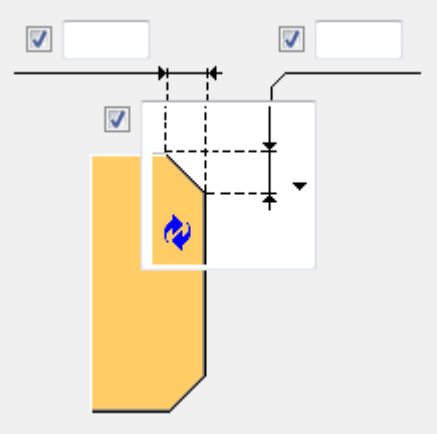
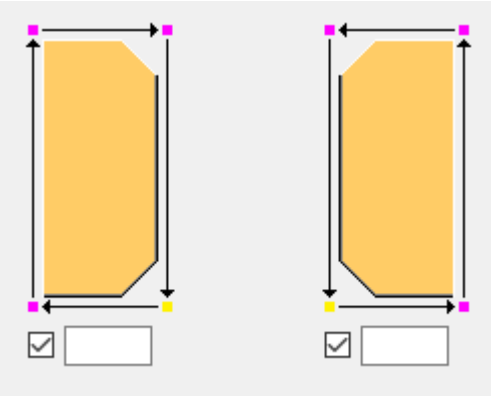
Use the **Parts** tab to control the stiffener properties.

Option	Description
Left stiffener **Right stiffener**	Define the stiffener thickness, width and height. By default, the thickness is `1.5*beam web thickness` rounded up to the next plate thickness.

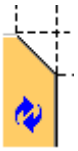
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Parameters tab

Use the **Parameters** tab to control the stiffener angle and chamfers.

Option	Description
	Position of the stiffeners. You can enter horizontal angles in degrees for the right and the left stiffener.
	Chamfer type and dimensions. If you select the **No Chamfer** option, a clash can occur between the stiffener and the I-profile rounding. Additionally, you can define the chamfer dimensions vertically and horizontally. If you select an arc chamfer, the horizontal dimension is the radius and the vertical dimension has no effect.
	Set the stiffener polygon rotation of the right and the left stiffener. The value you enter determines the number of points by which the start point of the polygon is moved in the direction shown by the arrows.

Chamfer type

Option	Description
	Default Line chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

General tab

Click the link below to find out more:

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

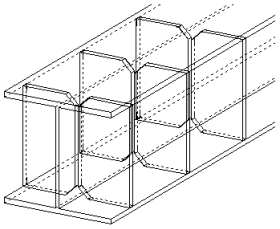
Multiple stiffeners (1064)

Multiple stiffeners (1064) creates multiple stiffeners to I, C, and U profiles.

Objects created

- Stiffeners
- Welds

Use for

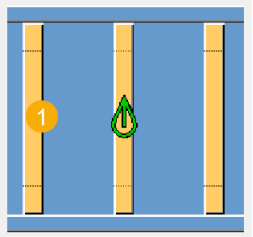
Situation	Description
	Multiple stiffeners created to an I beam.

Selection order

1. Select the main part (beam or column).
2. Pick a position for the stiffeners.

   The stiffeners are created automatically when you pick the position.

Part identification key

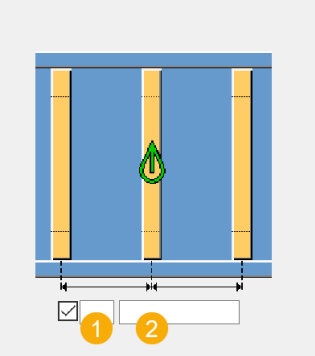


	Description
1	Stiffener

Picture tab

Use the **Picture** tab to define the number and spacing of the stiffeners.

Stiffeners



	Description
1	Number of stiffeners.
2	Stiffener spacing. Use a space to separate stiffener spacing values. Enter a value for each space between stiffeners. For example, if there are 3 stiffeners, enter 2 values.

Parts tab

Use the **Parts** tab to define the part properties.

Parts

Part	Description
Left stiffener	Thickness and width of the left stiffener.
Right stiffener	Thickness and width of the right stiffener.

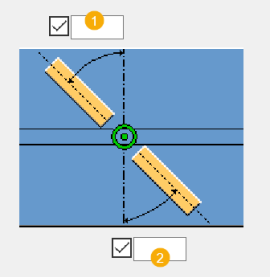
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in

Option	Description	Default
		File menu --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Parameters tab

Use the **Parameters** tab to define the stiffener angles, creation direction, and offset.

Stiffener angles



	Description	Default
1	Angle of the right stiffener.	0 degrees
2	Angle of the left stiffener.	0 degrees

Stiffener creation direction

Option	Description
	Default Middle AutoDefaults can change this option.
	Middle Creates stiffeners symmetrically.
	Right Creates stiffeners to the right of the up direction symbol.
	Left Creates stiffeners to the left of the up direction symbol.

Stiffener offset location

Option	Description
	Default Middle AutoDefaults can change this option.
	Middle Sets the offset to the middle of the stiffener.
	Left Sets the offset to the left side of the stiffener.
	Right Sets the offset to the right side of the stiffener.

Fit stiffeners with flanges

Select whether sloped stiffeners are fitted with the main part flanges.

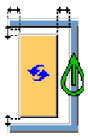
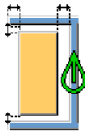
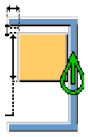
Apply gaps in the coordinate system of

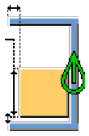
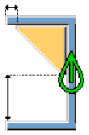
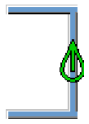
Select whether gaps are applied in the stiffeners or in the main part.

Stiffeners tab

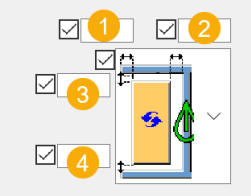
Use the **Stiffeners** tab to define stiffener creation, dimensions, and chamfers on both sides of the part.

Stiffener creation

Option	Description
	Default Full stiffener AutoDefaults can change this option.
	Full stiffener
	Partial stiffener, upper end

Option	Description
	Partial stiffener, lower end
	Partial stiffener, bevel
	No stiffeners are created.

Stiffener dimensions



	Option	Default
1	Stiffener edge distance from the beam edge.	0.0 mm
2	Stiffener edge distance from the beam web.	Beam rounding radius 1
3	Stiffener edge distance from the beam upper flange.	0 mm
4	Stiffener edge distance from the beam lower flange.	0 mm

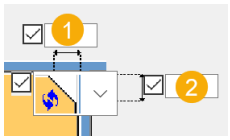
Chamfer type

You can select the chamfer type in each stiffener corner. The default type depends on the stiffener corner.

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer

Option	Description
	Convex chamfer
	Concave chamfer
	Square chamfer

Chamfer dimensions



	Description
1	Horizontal chamfer dimension.
2	Vertical chamfer dimension.

General tab

Click the link below to find out more:

General tab

Design type tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Standard gusset (1065)

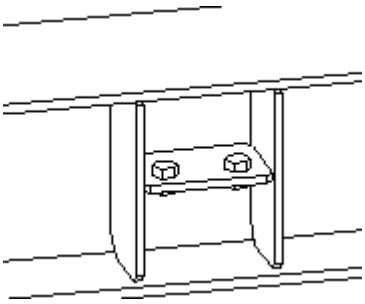
Standard gusset (1065) creates a gusset plate and welds it to the beam or column web. A stiffener is automatically created on each end of the gusset plate.

Objects created

- Gusset plate
- Stiffener plates (2)
- Bolts

To create braces and connect them to the gusset plate, use **Gusset tube in points (S47)**, **Crushed tube in bolts (S48)**, or **Gusset tube in bolts (S49)**.

Use for

Situation	Description
	Gusset plate welded to the beam web

Before you start

Create a beam or a column (H, W, or C profile).

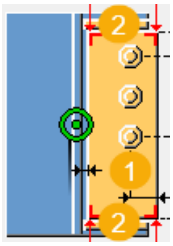
Note that the gusset plate is created using the properties you define on the **Parts** tab.

Selection order

1. Select the main part (beam or column).
2. Pick a point on the main part to indicate the location of the connection.

   The connection is created automatically when the you pick the point.

Part identification key

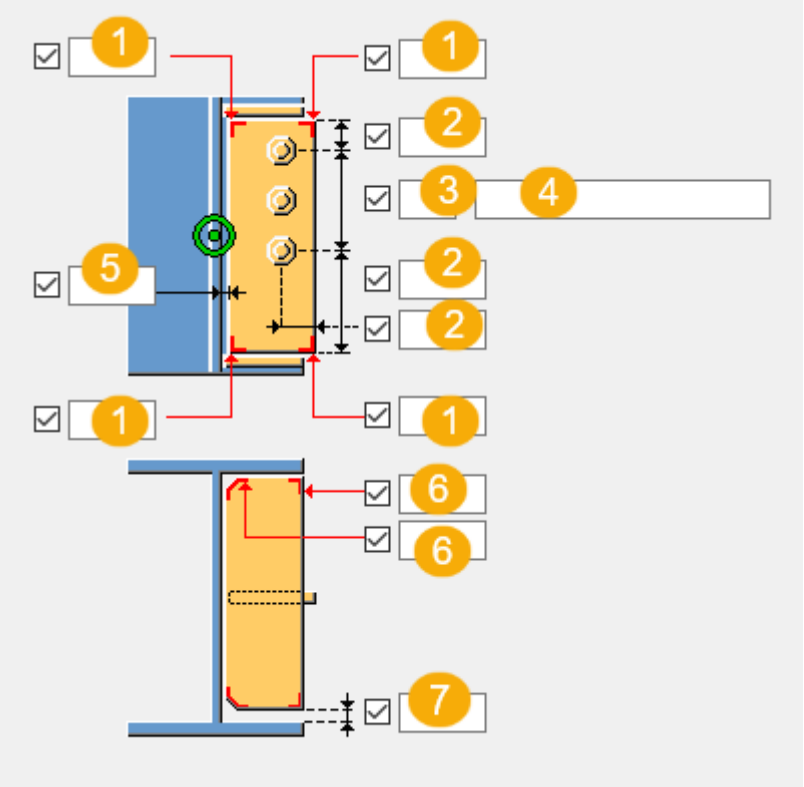


1	Gusset plate
2	Stiffener

Picture tab

Use the **Picture** tab to define the gusset plate and stiffener dimensions.

Part dimensions



	Description
1	Define the corner cut size of the gusset plate.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Define the gusset plate offset from the beam or column web.
6	Define the corner cut size of the stiffener.
7	Define the stiffener offset from the beam or column web.

Parts tab

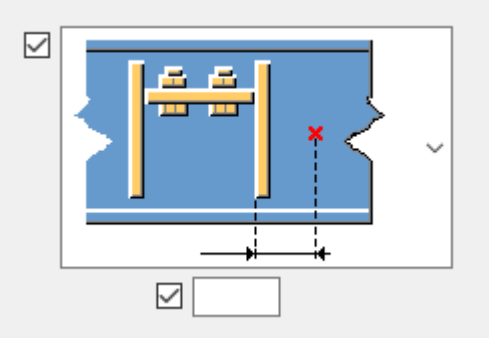
Use the **Parts** tab define the gusset plate and stiffener properties.

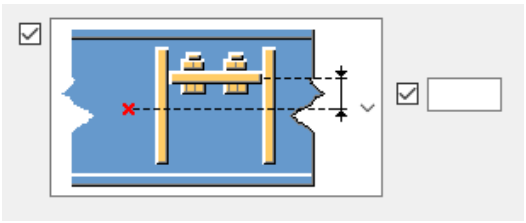
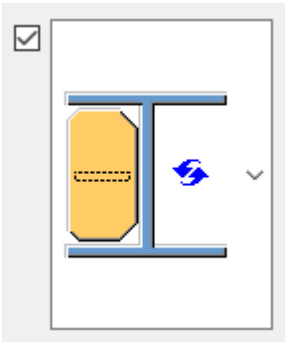
Part	Description
Gusset plate, **Front stiffener**, **Back stiffener**	Define the thickness, width and height of the gusset plate and the stiffeners.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

Use the **Parts** tab to define the position of the gusset.

Option	Description
	Define the horizontal gusset location using a reference point.

Option	Description
	Define the vertical gusset location using a reference point.
	Select the gusset location.

Bolts tab

Use the **Bolts** tab to define the bolt properties.

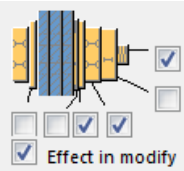
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Bolt group

Select whether several or a single bolt group is created.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

5.15 Base plates

This section introduces base plate components available in Tekla Structures.

Click the links below to find out more:

- U.S. Base plate connection (71) (page 2521)
- Round joining plates (124) (page 2543)
- Base plate (1004) (page 2550)

- Stiffened base plate (1014) (page 2576)
- Web stiffened base plate (1016) (page 2604)
- Simple base plate 2 (1031) (page 2627)
- Base plate (1042) (page 2642)
- U.S. Bearing plate (1044) (page 2667)
- U.S. Base plate (1047) (page 2677)
- Circular base plates (1052) (page 2706)
- Base plate (1053) (page 2725)
- Box column base plate (1066) (page 2733)

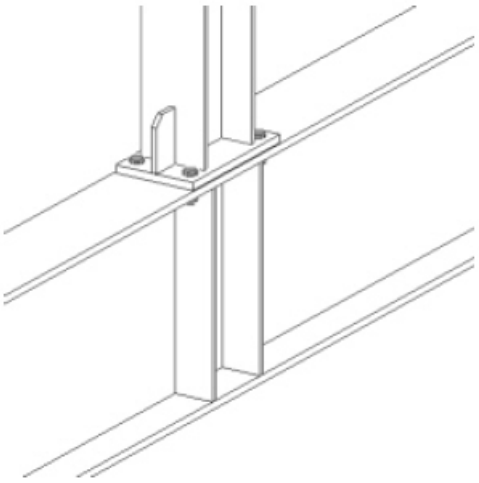
U.S. Base plate connection (71)

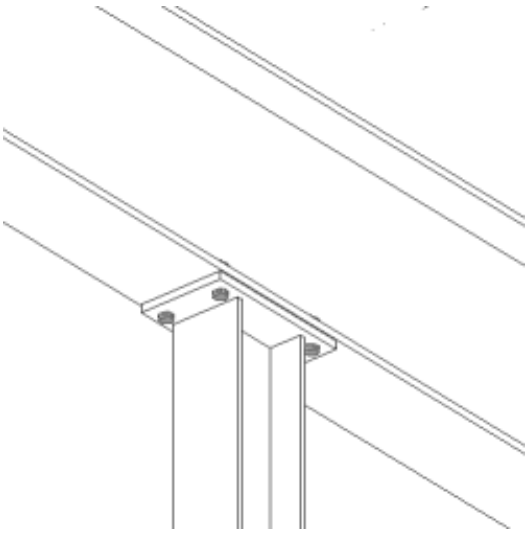
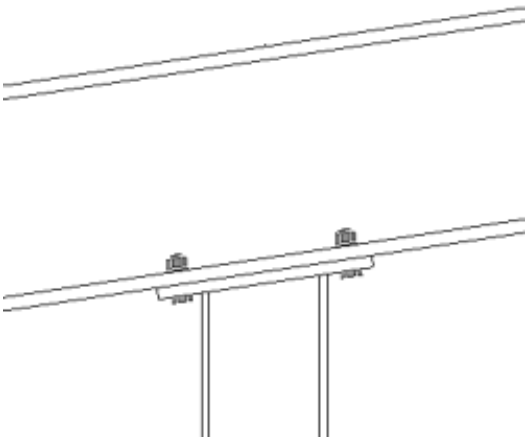
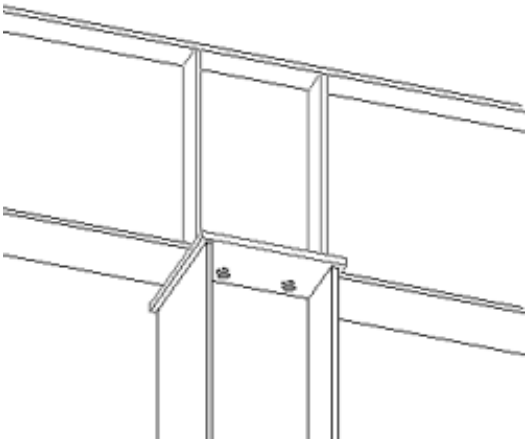
U.S. Base plate connection (71) creates a base plate that connects a column to a beam.

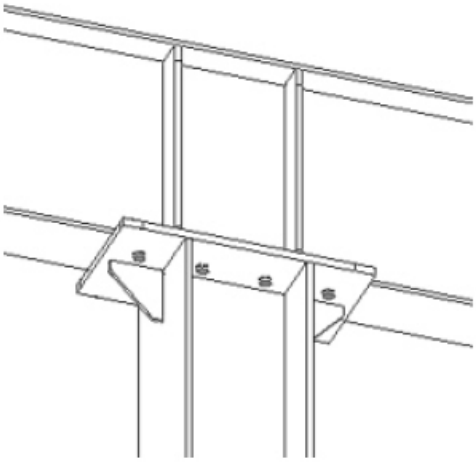
Objects created

- Base plate
- Stiffeners (can only be added if the column has a W, I, or a TS profile)
- Beam stiffener
- Bolts
- Welds

Use for

Situation	Description
	Base plate is connected to the beam. Beam and column stiffeners are used.

Situation	Description
	Beam cantilever is connected to a column with a base plate.
	Beam cantilever is connected to a sloped column with a base plate.
	Beam cantilever is connected to a column with a base plate. Beam stiffeners are used.

Situation	Description
	Beam cantilever is connected to the column with a base plate. Beam and column stiffener are used.

Before you start

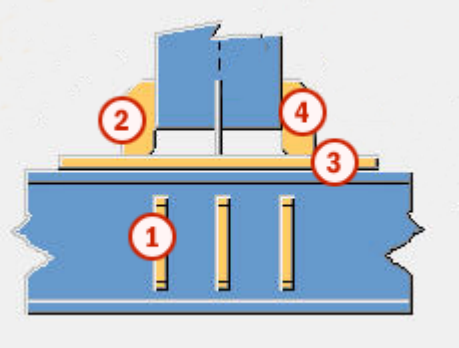
Create a beam and a column.

Selection order

1. Select the main part (beam).
2. Select the secondary part (column).

   The connection is created automatically.

Part identification key

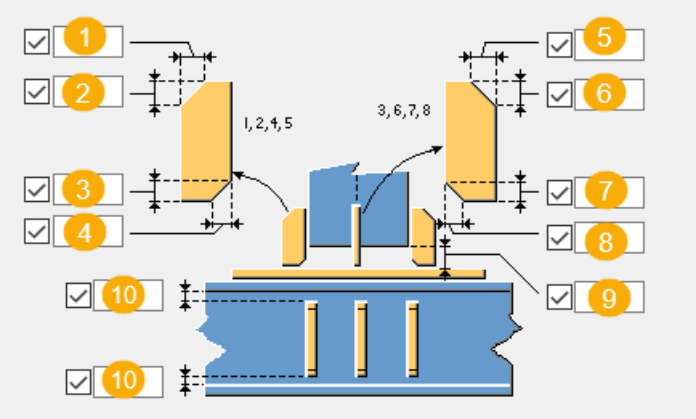


	Part
1	Beam stiffener
2	Stiffeners 1, 2, 4, 5
3	Base plate
4	Stiffeners 3, 6, 7, 8

Picture tab

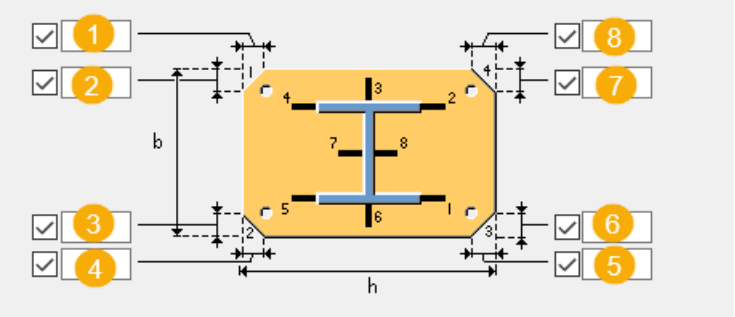
Use the **Picture** tab to control the stiffener chamfering and the base plate chamfers.

Stiffener chamfering



	Description
1	Top chamfer width for stiffeners 1, 2, 4, 5.
2	Top chamfer height for stiffeners 1, 2, 4, 5.
3	Bottom chamfer height for stiffeners 1, 2, 4, 5.
4	Bottom chamfer width for stiffeners 1, 2, 4, 5.
5	Top chamfer width for stiffeners 3, 6, 7, 8.
6	Top chamfer height for stiffeners 3, 6, 7, 8.
7	Bottom chamfer height for stiffeners 3, 6, 7, 8.
8	Bottom chamfer width for stiffeners 3, 6, 7, 8.
9	Weld gap.
10	Gap between the beam stiffeners and the beam top and bottom flange.

Base plate chamfering



	Description
1	Width of base plate chamfer 1.

	Description
2	Height of base plate chamfer 1.
3	Height of base plate chamfer 2.
4	Width of base plate chamfer 2.
5	Width of base plate chamfer 3.
6	Height of base plate chamfer 3.
7	Height of base plate chamfer 4.
8	Width of base plate chamfer 4.

Parts tab

Use the **Parts** tab to control the dimensions of the base plate, stiffeners, and beam stiffener.

Plate

Option	Description	Default
Plate	Base plate thickness, width and height.	Thickness = `20 mm` Width depends on the column profile. Height depends on the column profile. The default name is `BASEPLATE`.
Stiffeners 1,2,4,5	Stiffener 1, 2, 4, 5 thickness, width and height.	
Stiffeners 3, 6	Stiffener 3, 6 thickness, width and height.	
Stiffeners 7,8	Stiffener 7, 8 thickness, width and height.	
Beam Stiff	Beam stiffener thickness, width and height.	

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
	assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

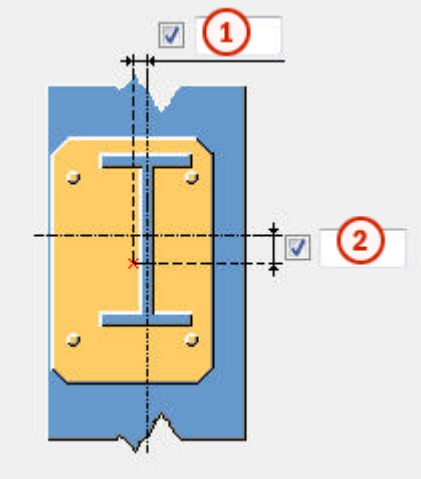
Parameters tab

Use the **Parameters** tab to control the square cut corners, plate offset and bolt properties.

Plate properties

Option	Description
Square cut corners (1-4)	Cuts made on the corners of the base plate. If you enter a number corresponding to one of the corners of the base plate, the corner has a square cut instead of a diagonal chamfer. You can enter the numbers for one or all the corners.
Plate aligned with	Define whether the plate is aligned with the column or the connection symbol.

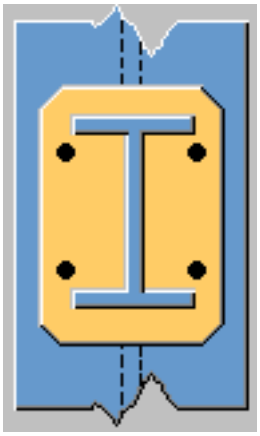
Base plate offset



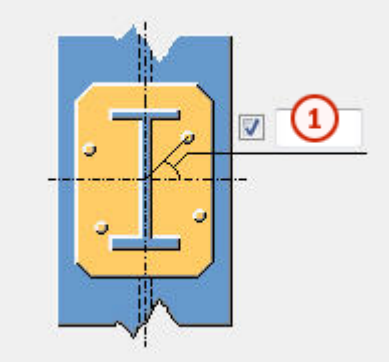
	Description
1	Base plate horizontal offset from the column center.
2	Base plate vertical offset from the column center.

Base plate with holes or bolts

Option	Description
	Default Bolts are created. AutoDefaults can change this option.

Option	Description
	Bolts are created.
	Holes are created.

Bolt group rotation



You can define the bolt group rotation around its center. To rotate the bolt group, enter the rotation angle (in degrees).

Positive angle rotates the bolts in a counter-clockwise direction and negative angle in a clockwise direction.

	Description
1	Bolt group rotation angle.

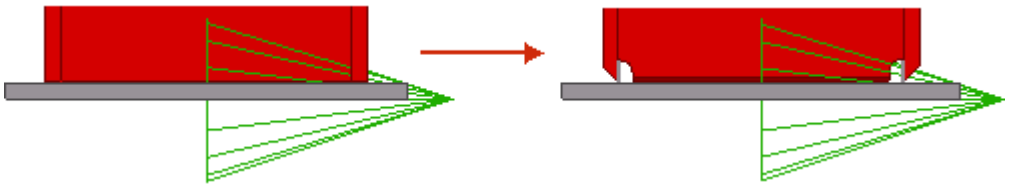
Bolt properties

Option	Description
Eliminate bolt numbers	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to down.
Eliminate external bolts	Define whether external bolts are deleted.

Using additional components

You can use additional system or custom components to modify the column end or the base plate. For example, you can create special backing plates, weld preparations, and weld access holes for the column end.

If you use additional system or custom components, you need to manage the column end or the base plate properties in the additional component in question. When using several components, there may be multiple welds and cuts.



Option	Description
Component	Define a system or a custom component by selecting it from the component catalog.
Attributes	Enter the name of the attribute file for the selected component.
Input	Define to which parts the selected component is applied. • **Default** is same as **Base + Column**. • **Column** sets the column as the main part. Use this option for details. • **Column + Base** sets the column as the main part and the base plate as the secondary part.

Option	Description
	• **Base + Column** sets the base plate as the main part and the column as the secondary part. • **Base** sets the base plate as the main part.

Stiffeners tab

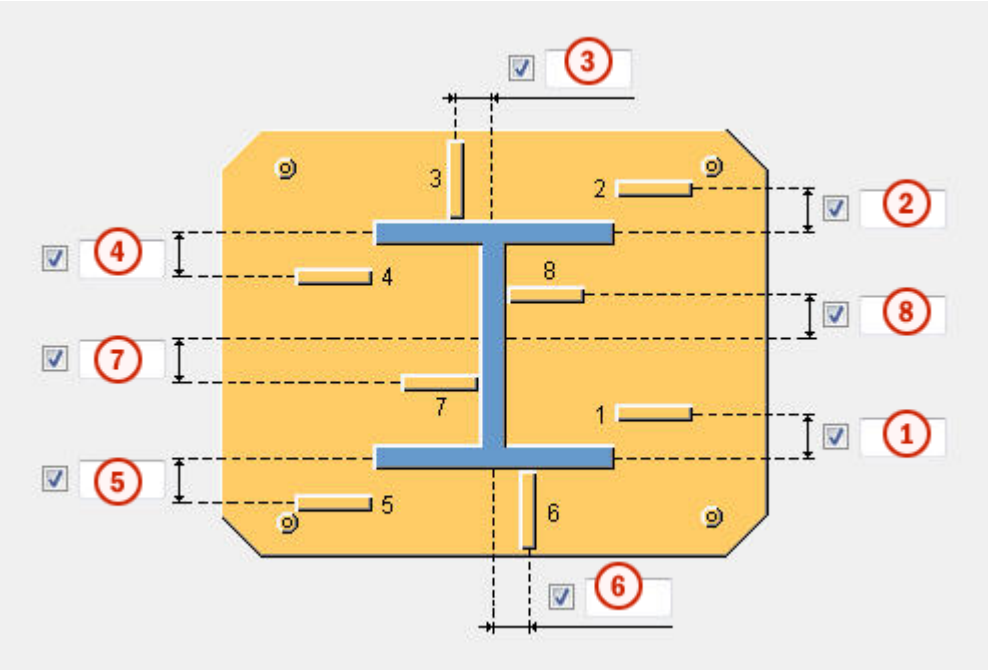
Use the **Stiffeners** tab to control stiffener positions and offsets.

Stiffener positions (1-8)

Option	Description
Stiffener Positions (1-8)	Placement of the stiffeners. Only the stiffeners whose numbers are entered in the field are attached to the column.

Stiffener offset

Stiffener offset allows the stiffeners to be moved. Enter a negative value to move the stiffener in the opposite direction.



	Description
1	Stiffener 1 offset.
2	Stiffener 2 offset.

	Description
3	Stiffener 3 offset.
4	Stiffener 4 offset.
5	Stiffener 5 offset.
6	Stiffener 6 offset.
7	Stiffener 7 offset.
8	Stiffener 8 offset.

Beam Stiff tab

Use the **Beam Stiff** tab to control the beam stiffener alignment, stiffener side, chamfer dimensions and types.

Left stiffener

Option	Description
	Default Left stiffeners are created. AutoDefaults can change this option.
	Left stiffeners are created.
	Left stiffeners are not created.

Center stiffener

Option	Description
	Default Center stiffeners are created. AutoDefaults can change this option.
	Center stiffeners are created.
	Center stiffeners are not created.

Right stiffener

Option	Description
	Default Right stiffeners are created. AutoDefaults can change this option.

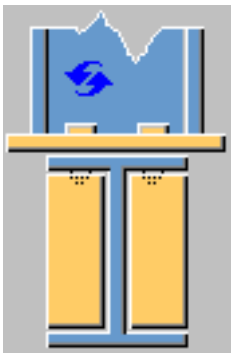
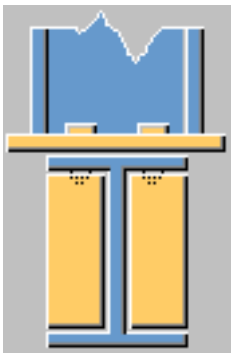
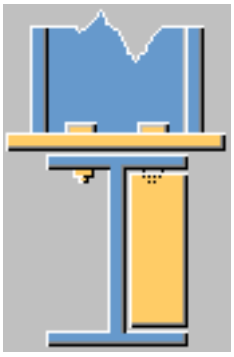
Option	Description
	Right stiffeners are created.
	Right stiffeners are not created.

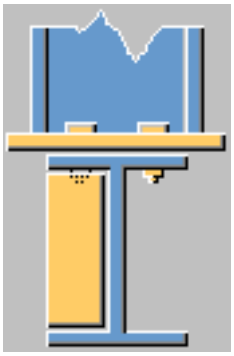
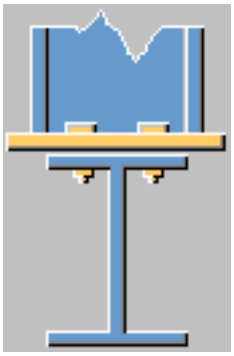
Stiffener alignment

Options	Description
	Default Stiffeners are aligned with the beam. AutoDefaults can change this option.
	Stiffeners are aligned with the beam.

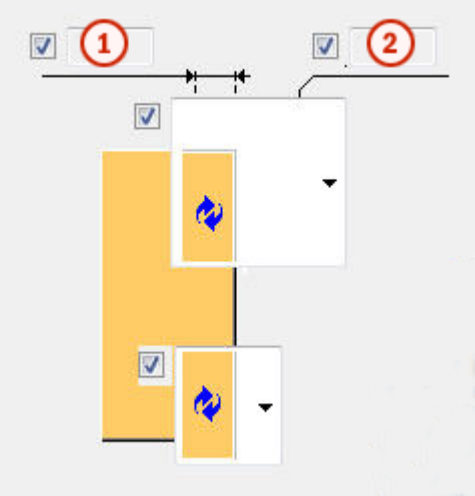
Options	Description
	Stiffeners are aligned with the column.

Stiffener creation

Option	Description
	Default Stiffeners are created on both sides. AutoDefaults can change this option.
	Stiffeners are created on both sides.
	Stiffeners are created on far side.

Option	Description
	Stiffeners are created on near side.
	Stiffeners are not created.

Chamfer dimensions



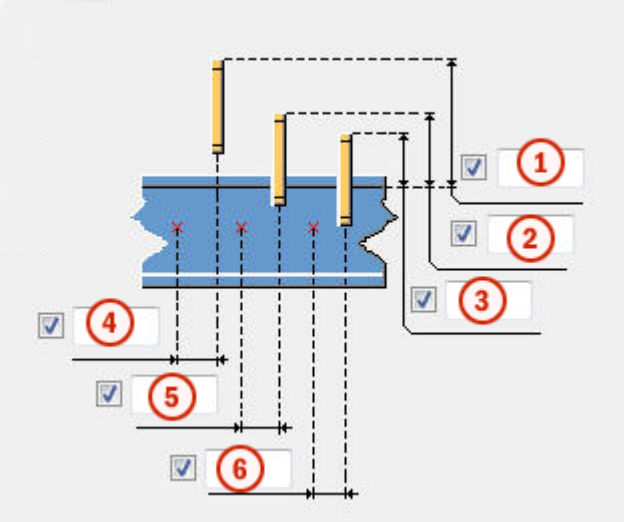
	Description
1	Horizontal dimension of the chamfer.
2	Vertical dimension of the chamfer.

Top and bottom chamfer type

Option for top chamfer	Option for bottom chamfer	Description
		Default No chamfer AutoDefaults can change this option.
		No chamfer
		Line chamfer
		Convex arc chamfer
		Concave arc chamfer

Stiffener offset

Stiffeners can be moved from the center of the beam horizontally or vertically. Enter a negative value to move the stiffeners in the opposite direction.

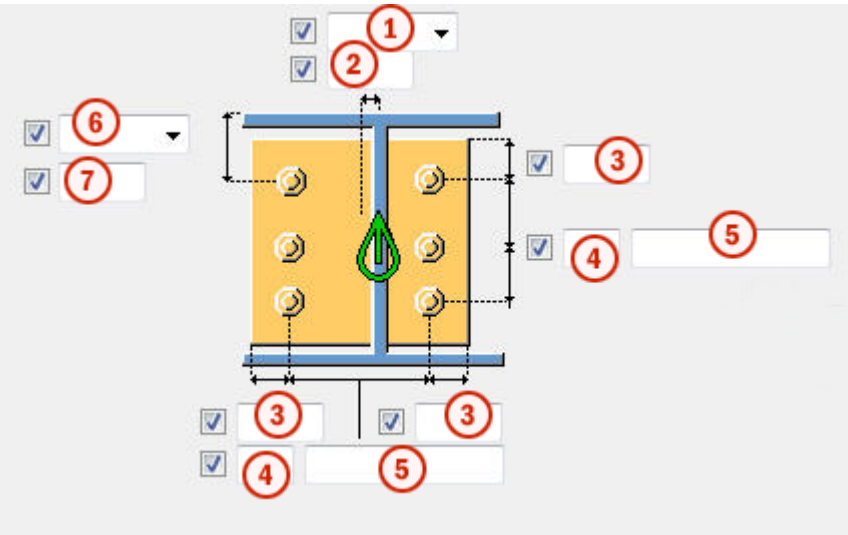


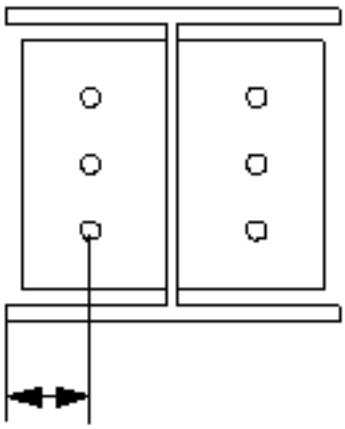
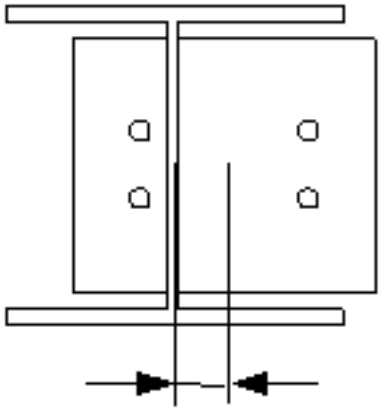
	Description
1	Left stiffener vertical offset.
2	Center stiffener vertical offset.
3	Right stiffener vertical offset.
4	Left stiffener horizontal offset.
5	Center stiffener horizontal offset.
6	Right stiffener horizontal offset.

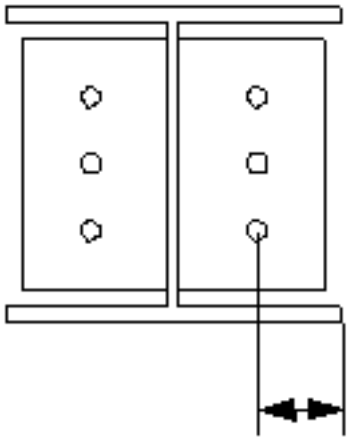
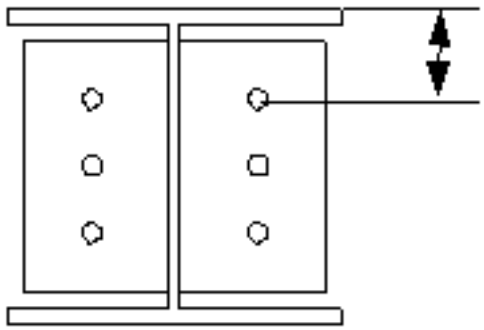
Bolts tab

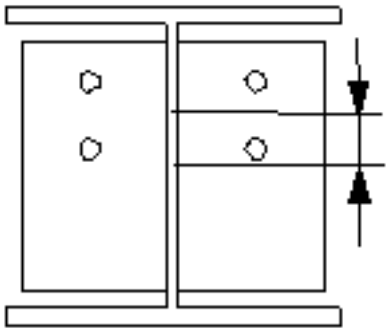
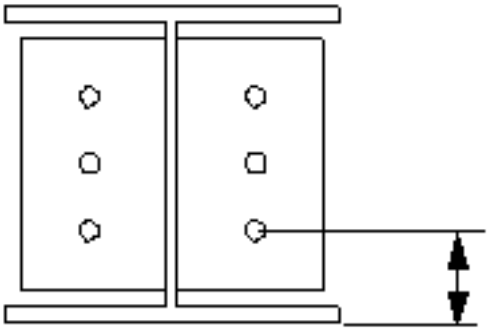
Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt.  • **Middle**: From the center line of the secondary part to the center line of the bolts. 

	Description
	• **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Dimension for vertical bolt group position.

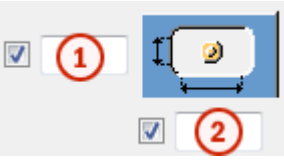
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

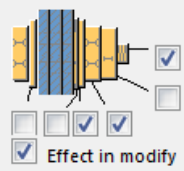


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

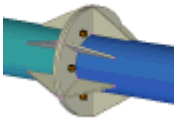
Round joining plates (124)

Round joining plates (124) connects two beams or two columns with bolted base plates. Stiffeners can also be created.

Objects created

- Base plates
- Stiffeners
- Bolts
- Welds

Use for

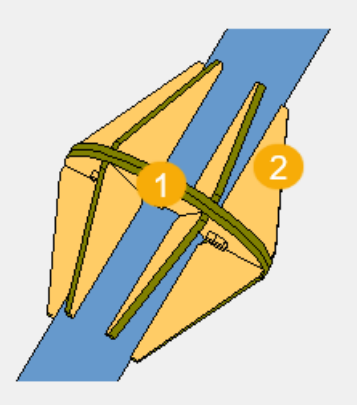
Situation	Description
	Round end plates connecting two beams. Stiffeners are created.

Selection order

1. Select the main part.
2. Select the secondary part.

   The connection is created automatically when the secondary part is selected.

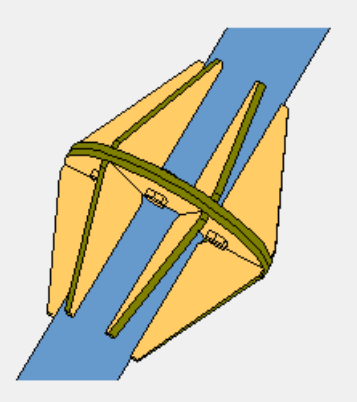
Part identification key



	Description
1	Base plate
2	Tube stiffener

Picture tab

Joining plates



Parts tab

Use the **Parts** tab to control the base plate and stiffener properties.

Parts

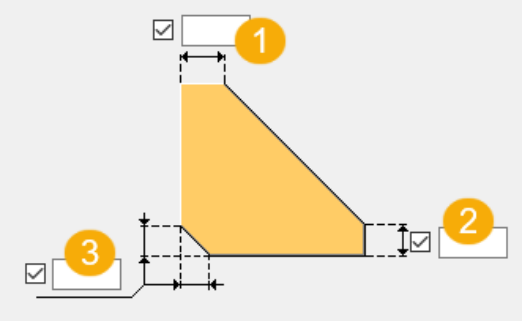
Option	Description
Base plate	Thickness and width of the base plate.
Tube stiffeners 1, **Tube stiffeners 2**	Thickness, width and height of the tube stiffeners.

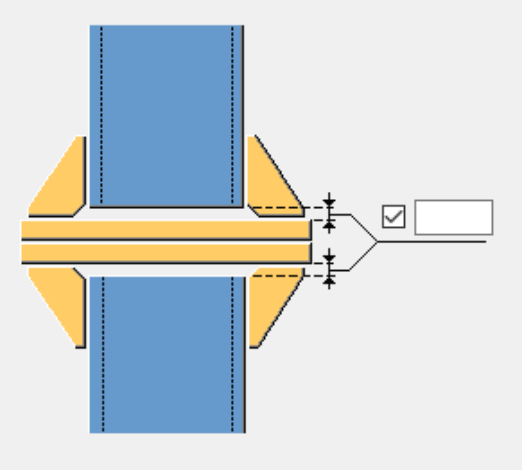
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Parameters tab

Use the **Parameters** tab to control the shape and position of the base plate, the number of stiffeners, and the chamfer dimensions.

Stiffeners

Option	Description
Quantity of stiffeners	Define how many stiffeners are created.
	Define the stiffener chamfer dimensions: 1. Chamfers the stiffener corner where it meets only the column. 2. Chamfers the stiffener corner at the outer edge of the base plate. 3. Chamfers the stiffener corner at the intersection of the column and the base plate.

Option	Description
	Define the weld gap between the parts and the base plates.

Base plate shape

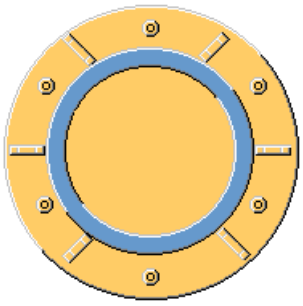
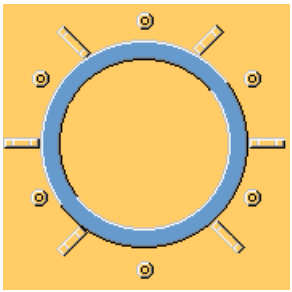
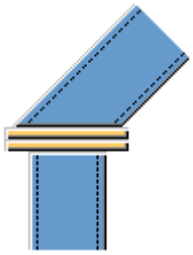
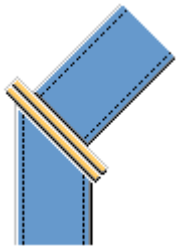
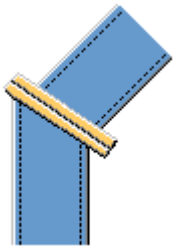
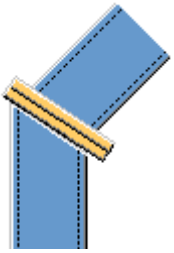
Option	Description
	Round base plate
	Square base plate

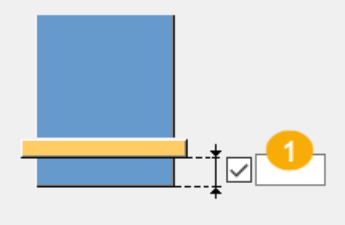
Plate type

Select the plate type.

Base plate direction

Option	Description
	Creates the base plates perpendicular to the main part.
	Creates the base plates perpendicular to the secondary part.
	Creates the base plates at an angle which bisects the angle of the main and secondary parts.
	Creates the base plates parallel with the connection symbol.

Overlap

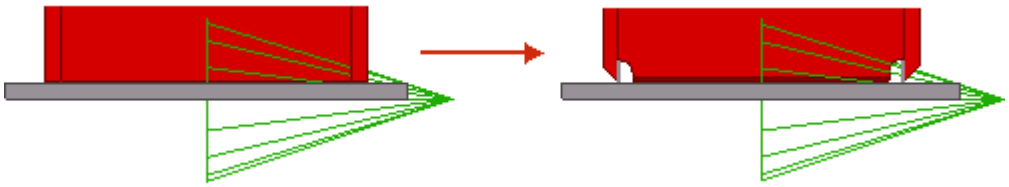


	Description
1	Distance between the part flange and the plate flange.

Using additional components

You can use additional system or custom components to modify the column end or the base plate. For example, you can create special backing plates, weld preparations, and weld access holes for the column end.

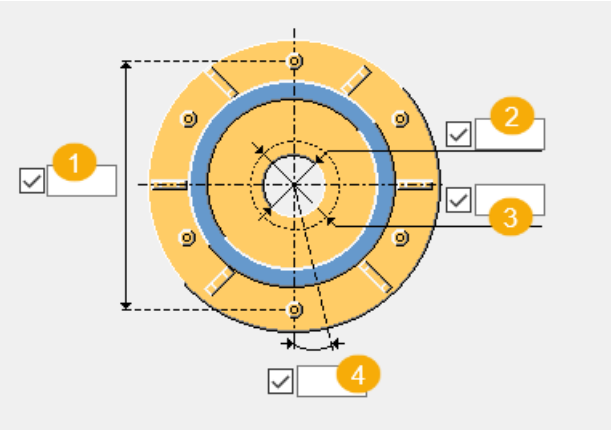
If you use additional system or custom components, you need to manage the column end or the base plate properties in the additional component in question. When using several components, there may be multiple welds and cuts.



Option	Description
Component	Define a system or a custom component by selecting it from the component catalog.
Attributes	Enter the name of the attribute file for the selected component.
Input	Define to which parts the selected component is applied. • **Default** is the same as **Base + Column**. • **Column** sets the column as the main part. Use this option for details. • **Column + Base** sets the column as the main part and the base plate as the secondary part. • **Base + Column** sets the base plate as the main part and the column as the secondary part. • **Base** sets the base plate as the main part.

Bolts tab

Use the **Bolts** tab to control the bolt properties, the diameter of the inner plate, and the bolt angle.



	Description	Default
1	Diameter of the bolt circle	
2	Base plate 1 inner diameter Creates a centered hole in base plate 1	Main part diameter plus tolerance
3	Base plate 2 inner diameter Creates a centered hole in base plate 2	Secondary part diameter plus tolerance
4	Bolt angle (in degrees)	

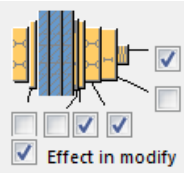
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

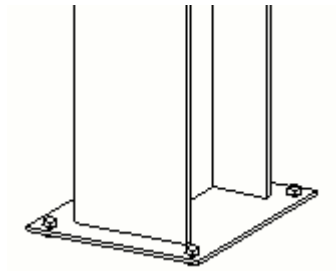
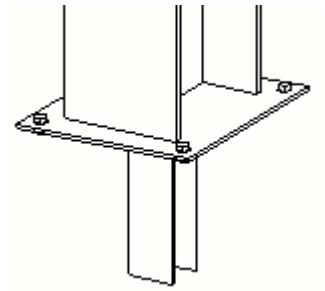
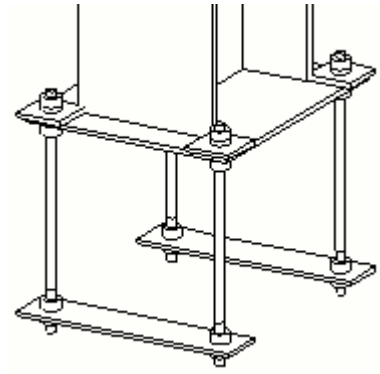
Base plate (1004)

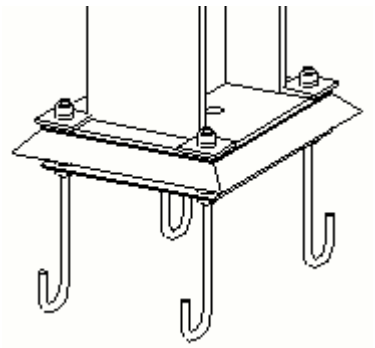
Base plate (1004) creates a base plate that is connected to a column end.

Objects created

- Base plate
- Shim plates (optional)
- Leveling plate (optional)
- Grout (optional)
- Shear key (optional)
- Extra plates connecting the anchor rods
- Anchor rods
- Bolts
- Additional component (optional)
- Welds

Use for

Situation	Description
	Simple base plate detail
	Base plate detail with a shear key
	Base plate detail with • Straight anchor rods • Extra plates connecting the anchors

Situation	Description
	Base plate detail with • Anchor rods with hooks • Leveling plate below the base plate • Grout and hole for grouting • Cast plate

Before you start

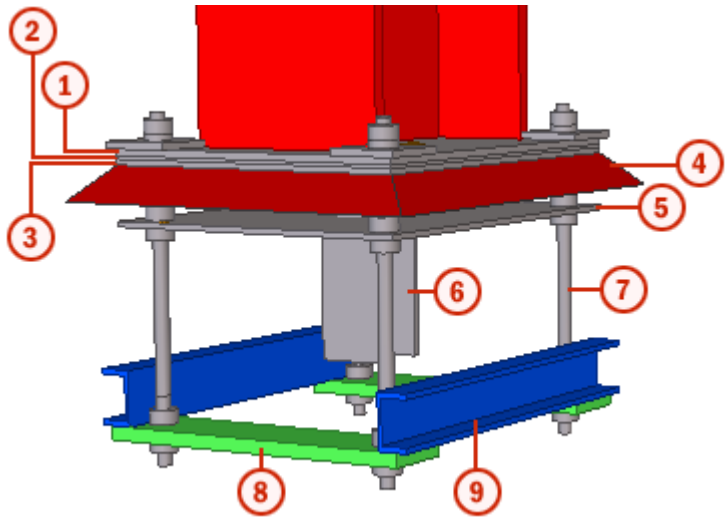
Create a column.

Selection order

1. Select the main part (column).
2. Pick a position.

   The detail is created automatically.

Part identification key



	Part
1	Base plate
2	Shim plate
3	Leveling plate
4	Grout
5	Cast plate
6	Shear key

	Part
7	Anchor rod
8	Extra plate 1
9	Extra plate 2

Example: Add a base plate and anchor rods using Base plate (1004)

In this example, you will add a base plate detail and anchor rods to a column.

1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. Enter `base plate` in the search box.

   To view the thumbnail images of the components in the search results click .
3. Select **Base plate (1004)**.
4. Select the column.
5. Pick a position at the base of the column.

   Tekla Structures automatically adds the base plate when you pick the position.
6. Next, modify the anchor rod dimensions.

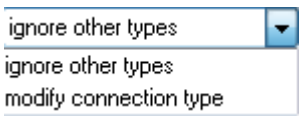
   a. Switch on the **Select components** switch to more easily select components.

   b. Double-click the component symbol in the model to open the **Base plate (1004)** component dialog box.

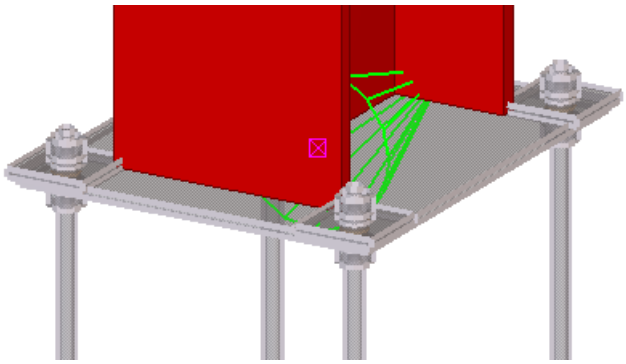
   c. Go to the **Anchor rods** tab.

   d. Change the dimensions of the anchor rods.

   e. To change only this base plate, select **ignore other types** from the list in the top part of the dialog box.

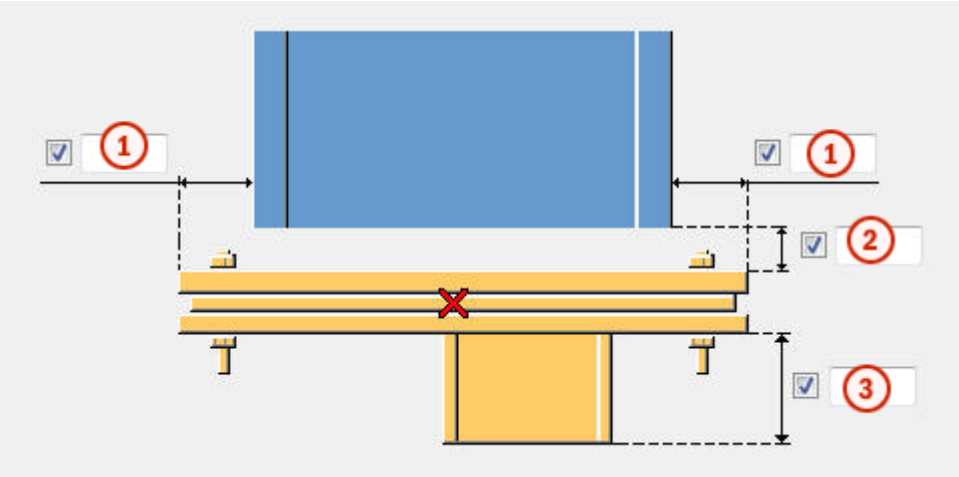
   f. Click **Modify**.



Picture tab

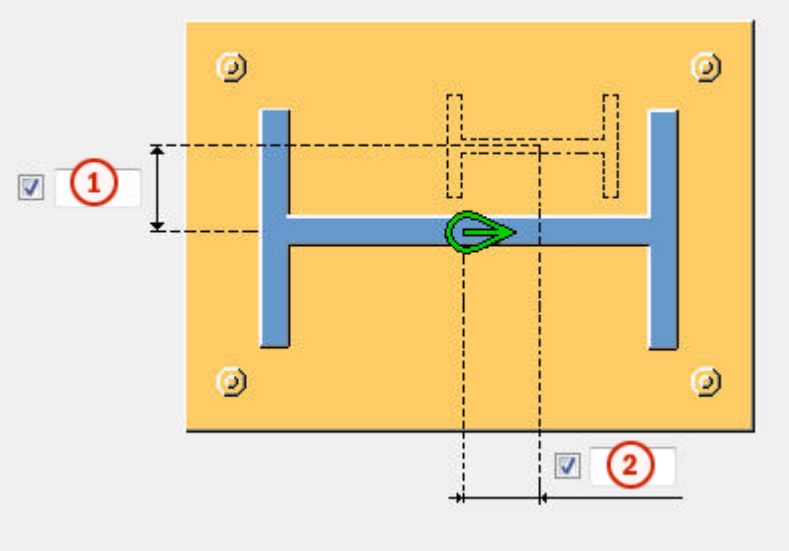
Use the **Picture** tab to control the position of the base plate and the length and position of the shear key.

Dimensions



	Description
1	Distance from the flange to the edge of the base plate. Enter a negative value to make the base plate larger.
2	Weld gap.
3	Height of the shear key.

Shear key offset



	Description
1	Shear key vertical offset from the column center.
2	Shear key horizontal offset from the column center.

Parts tab

Use the **Parts** tab to control the dimensions of the base plate, shear key, leveling plate, and shim plates.

Plate

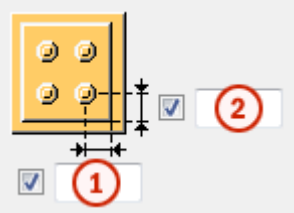
Option	Description	Default
Plate	Base plate thickness.	thickness = `0.5*bolt diameter rounded up to the next plate thickness` The default name is `BASEPLATE`.
Leveling Plate	Leveling plate thickness, width and height.	thickness = `1/4`
Leveling plate hole diameter	Leveling plate hole diameter.	
Leveling plate hole tolerance	Leveling plate hole tolerance.	

Option	Description	Default
Fitting plate	Shim plate thickness, width and height. Define up to three different shim plates.	
Number of fitting pl.	Number of shim plates for each thickness.	`1`
Additional beam	Shear key profile by selecting it from the profile catalog.	`HEA 300` (in Default environment)
Additional beam rotation	Select the shear key rotation type and define the rotation angle.	
Additional beam welded to	Define to which plate the shear key is welded.	

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

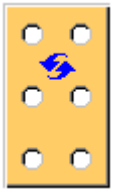
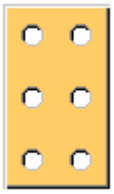
Bolt edge distances in shim plate

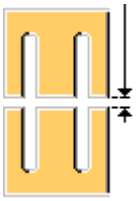
Define the bolt edge distances for shim plates. When these fields are empty, shim plates are of the same size as the base plate.



	Description	Default
1	Horizontal bolt edge distance in the shim plate.	30 mm
2	Vertical bolt edge distance in the shim plate.	30 mm

Shim plate shape

Option	Description
	Default Holes are based on the bolt group of the connection. AutoDefaults can change this option.
	Holes are based on the bolt group of the connection.
	Finger shim plate with horizontal slots. The plate can be installed from the right or the left side of the connection.
	Finger shim plate with vertical slots. The plate can be installed from the top of the connection.

Option	Description
	Two separate finger shim plates with horizontal slots.
	Two separate finger shim plates with vertical slots.

Tolerance

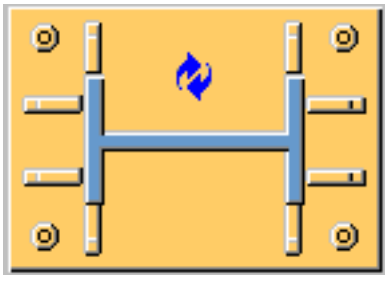
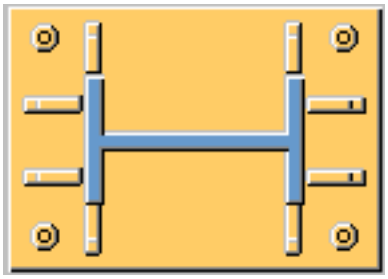
Define the tolerance for the slots in finger shim plates. The width of the slot is the bolt diameter + the tolerance.

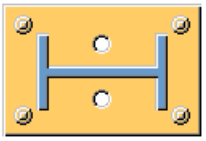
Parameters tab

Use the **Parameters** tab to control the component and the grout hole.

Grout hole

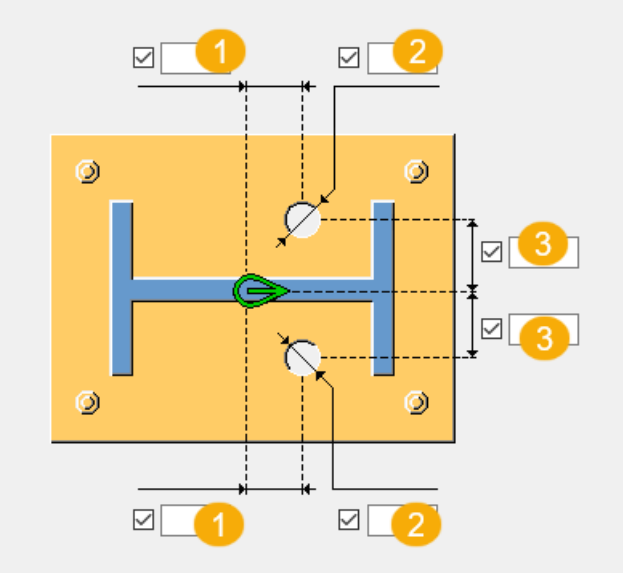
Define whether a grout hole is created in the base plate. The hole is also created in the leveling plate and shim plates, if they exist in the detail.

Option	Description
	Default Grout hole is not created. AutoDefaults can change this option.
	Grout hole is not created.

Option	Description
	Grout hole is created.
	Two grout holes are created.

Grout hole dimensions

If the base plate has two grout holes, define the dimensions for both holes.



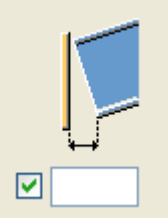
	Description
1	Location of the grout hole from the center of the column in the direction of the web.
2	Grout hole diameter.
3	Location of the grout hole from the center of the column in the direction of the flange.

Gap size

Define the limit value for the gap between the base plate and the column. Use this when the column is slightly inclined.

If the actual gap is smaller than this value, the end of the column is left straight.

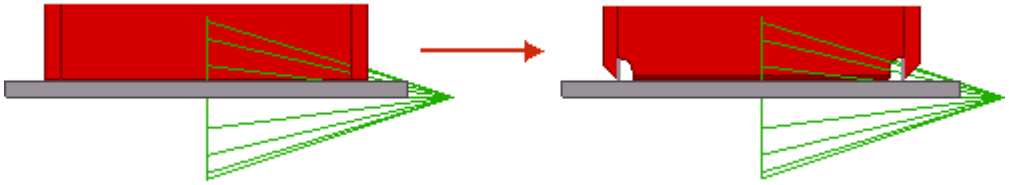
If the actual gap is larger than this value, the end of the column is fitted to the base plate.



Using additional components

You can use additional system or custom components to modify the column end or the base plate. For example, you can create special backing plates, weld preparations, and weld access holes for the column end.

If you use additional system or custom components, you need to manage the column end or the base plate properties in the additional component in question. When using several components, there may be multiple welds and cuts.

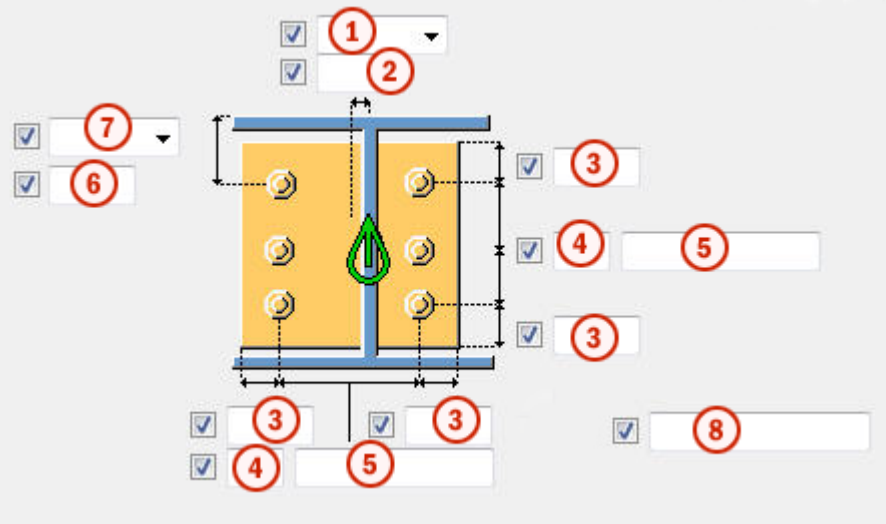


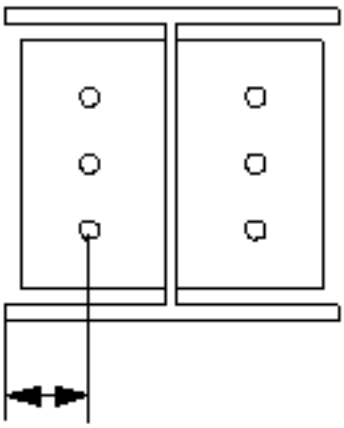
Option	Description
Component	Define a system or a custom component by selecting it from the component catalog.
Attributes	Enter the name of the attribute file for the selected component.
Input	Define to which parts the selected component is applied. • **Default** is same as **Base + Column**. • **Column** sets the column as the main part. Use this option for details. • **Column + Base** sets the column as the main part and the base plate as the secondary part. • **Base + Column** sets the base plate as the main part and the column as the secondary part. • **Base** sets the base plate as the main part.

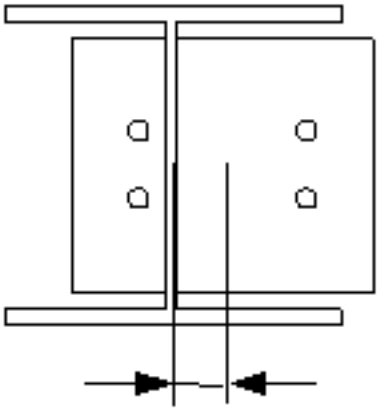
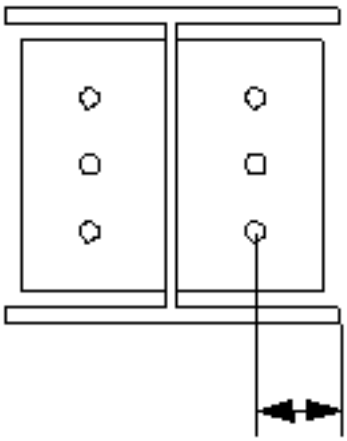
Bolts tab

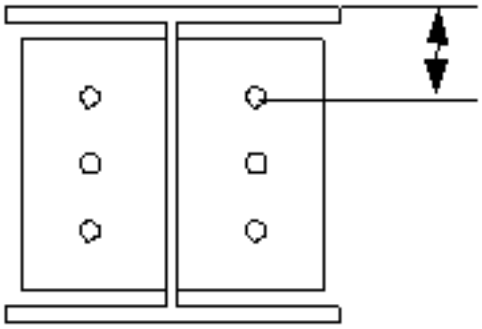
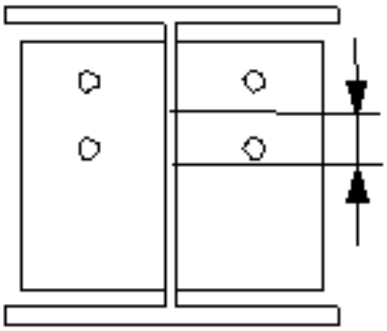
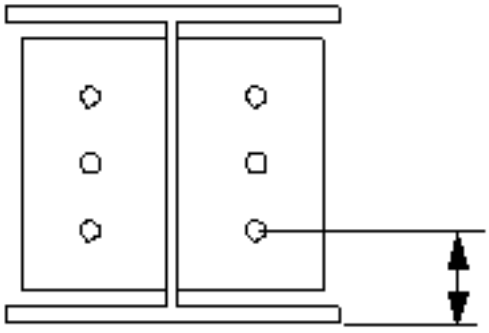
Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt. 

	Description
	• **Middle**: From the center line of the secondary part to the center line of the bolts.  • **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Dimension for vertical bolt group position.

	Description
7	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

	Description
8	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Bolt basic properties

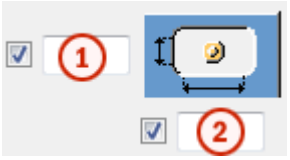
Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Slotted holes

You can define slotted, oversized, or tapped holes.



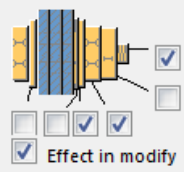
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.

Option	Description	Default
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Anchor rods tab

Use the **Anchor rods** tab to control the creation of different types of anchor rods.

Anchor rod dimensions

Option	Description
Rod profile	Anchor rod profile. You can add a comment about the part.
Nut profile	Nut profile.
Washer profile	Washer profile.
Plate washer	Plate washer thickness, width and height.
Cast plate	Cast plate thickness, width and height.
Grout	Grout thickness. Grouting helps you to model columns to the top of concrete parts and place the base plate correctly. It also makes it easier to dimension the detail in GA drawings. By default, no grouting is created. Select whether the grouting is created with or without slopes above or below the detail creation point. This also affects the shim plates.

Anchor rod part properties

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

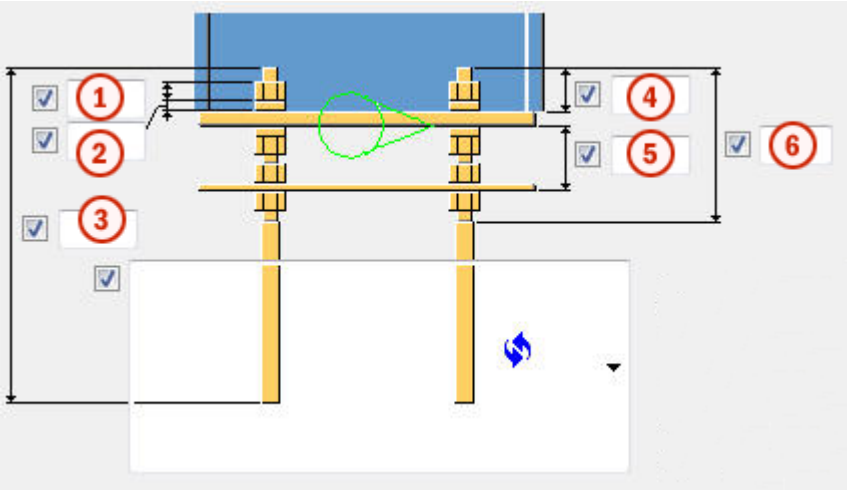
Option	Description	Default
Finish	Describes how the part surface has been treated.	

Base plate with

Select whether to create the base plate with bolts, anchor rods, or a custom component.

By default, the base plate is created with **Bolts**.

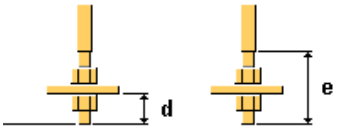
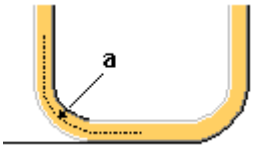
Anchor rod dimensions



	Description	Default
1	Size or the length of the nut.	`anchor rod diameter`
2	Size or the thickness of the washer.	`half of nut size`
3	Length of the anchor rod.	`500 mm`
4	Length of the anchor rod above the base plate.	`50 mm`
5	Distance between the cast plate and the base plate.	`60 mm`
6	Length of the upper thread.	`0 mm`

Anchor rods types

Option	Description	
	Default Type 1 AutoDefaults can change this option.	

Option	Description	
	Type 1	
	a Radius of the hook **b** Width of the hook	**a** = `2*anchor bar diameter` **b** = `1/5 of anchor bar length`
	a Radius of the hook **b** Width of the hook **c** Height of the hook	**c** = `same as width of the hook`
	**d** Length of the anchor rod below the extra plate **e** Length of the lower thread	**d** = `2*nut size` **e** = `4*nut size plus thickness of extra plate`
	**a** U-shape anchor rod Radius of the hook	

Hook direction

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1

Option	Description
	Type 2
	Type 3
	Type 4

Bolting direction

NOTE You can define the bolting direction if you have created the base plate with bolts.

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1
	Bolting direction 2

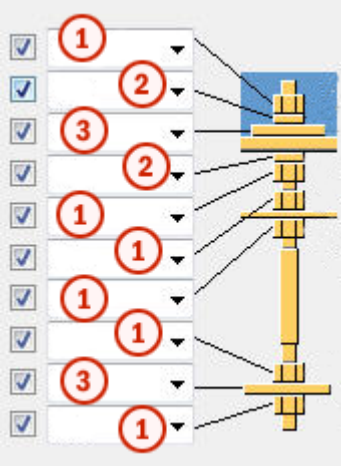
Cast plate holes tolerance

Option	Description	Default
Cast plate holes tolerance	Tolerance of the cast plate holes.	`same as bolt tolerance`

Washer hole tolerance

Option	Description
Create hole in washer	By default, a hole is not created in the washer. Tolerance of the washer hole.

Create



	Description
1	Create the nut profile.
2	Create the washer profile.
3	Create the washer plate.

Anchor rod assembly

Define which parts of the anchor rod are included in the anchor rod assembly.

You can weld the washer plates above and below the base plate.

Extra plates tab

Use the **Extra plates** tab to control the placement, rotation, and type of the profiles (extra profile 1) created at the bottom of each anchor bar and the profiles (extra profile 2) that connect rows of anchor bars.

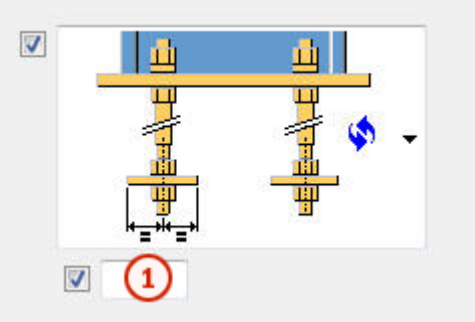
Part dimensions

Option	Description	Default
Extra profile 1	Define the first extra profile by selecting it from the profile catalog.	`PL10*100`
Extra profile 2	Define the second extra profile by selecting it from the profile catalog.	

Part properties

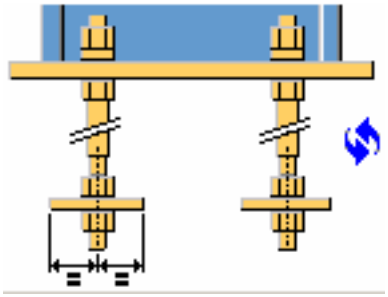
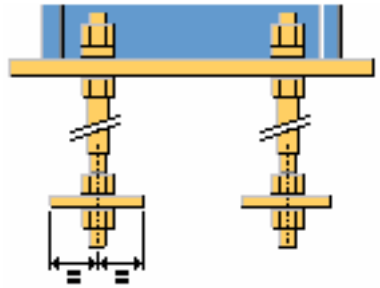
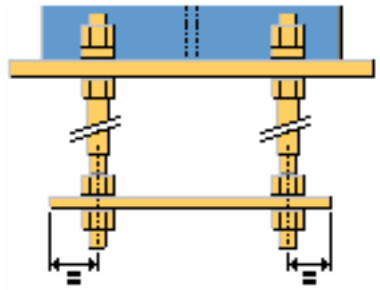
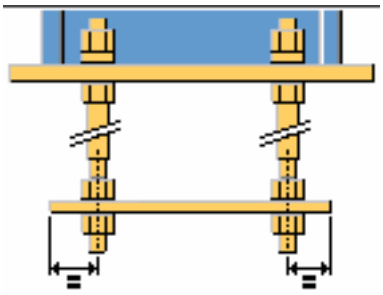
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Edge distance of extra profile 1

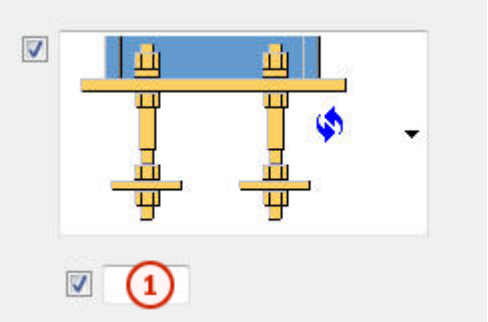


	Description	Default
1	Edge distance of extra profile 1.	50 mm

Type and direction of extra profile 1

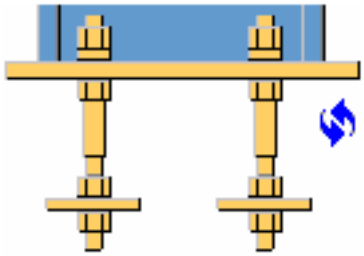
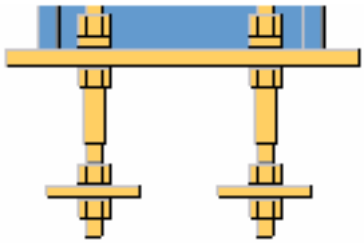
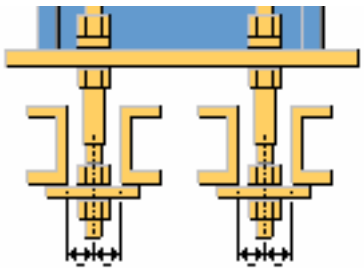
Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2
	Type 3

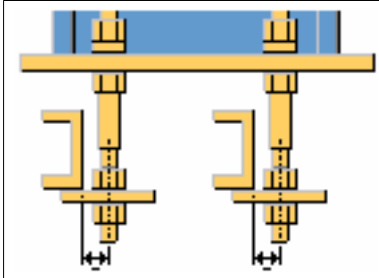
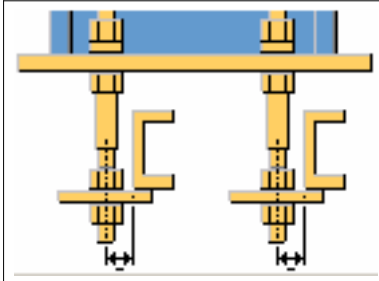
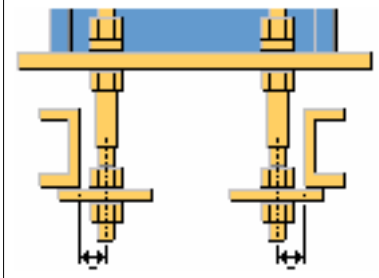
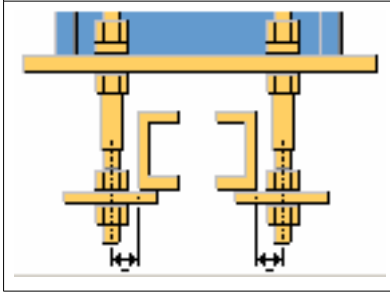
Edge distance of extra profile 2



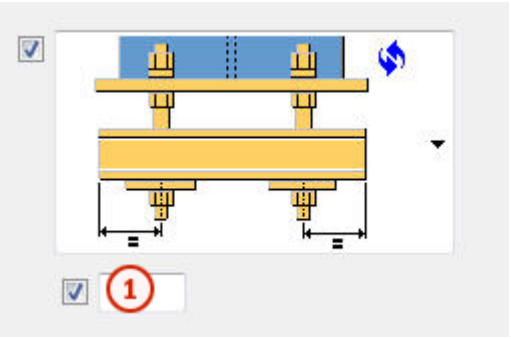
	Description	Default
1	Distance of extra profile 2 from the axis of the anchor bar.	`Half of the nut size or anchor bar diameter`

Extra profile 2 type

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2

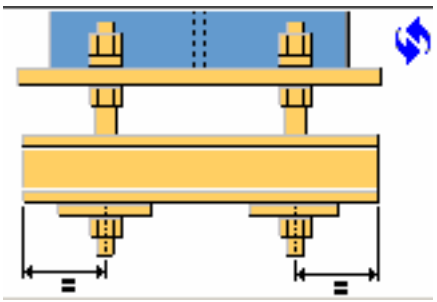
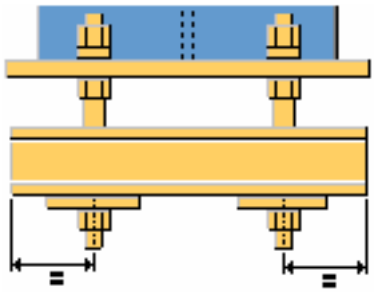
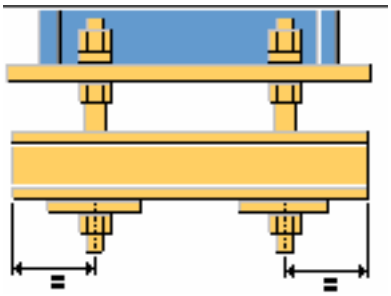
Option	Description
	Type 3
	Type 4
	Type 5
	Type 6

Length of extra profile 2



	Description	Default
1	Length of extra profile 2 from the axis of the anchor bar.	`50 mm`

Direction of extra profile 2

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2

Extra profile 1 properties

Option	Description	Default
Hole tolerance	Hole tolerance of extra profile 1.	`Same as bolt tolerance`
Circular profile height	Enter the height of a circular extra profile 1.	
Profile rotation	Profile rotation of extra profile 1.	`Front`

Extra profile 2 rotation

Option	Description	Default
Extra profile 2 rotation	Profile rotation of extra profile 2.	`Front`

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

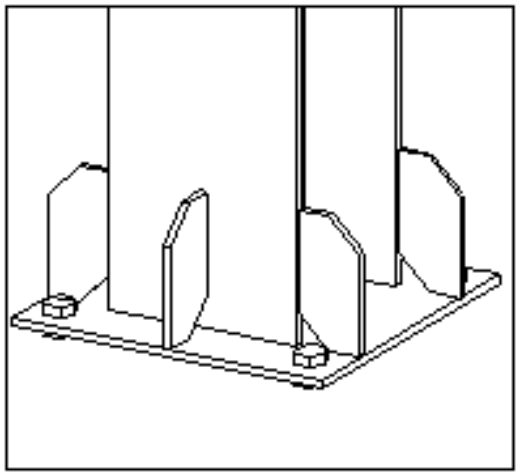
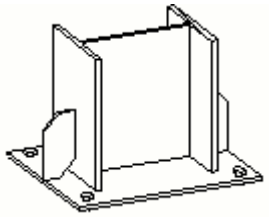
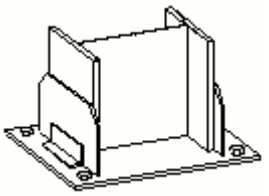
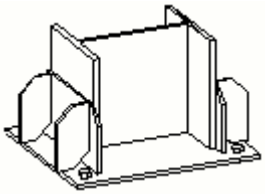
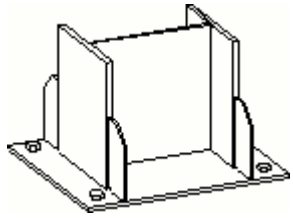
Stiffened base plate (1014)

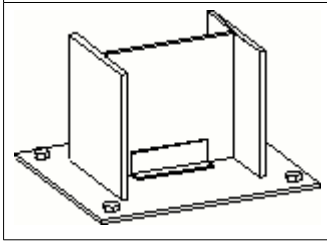
Stiffened base plate (1014) creates a base plate that is connected to a column end.

Objects created

- Base plate
- Stiffeners
- Shim plates (optional)
- Leveling plate (optional)
- Shear key (optional)
- Extra plates connecting the anchor rods
- Anchor rods
- Bolts
- Welds
- Additional component (optional)

Use for

Situation	Description
	Base plate with stiffeners
	Base plate with stiffeners on the flange
	Base plate with stiffeners on the flange
	Base plate with stiffeners on the flange
	Base plate with stiffeners on the flange

Situation	Description
	Base plate with stiffeners on the web

Before you start

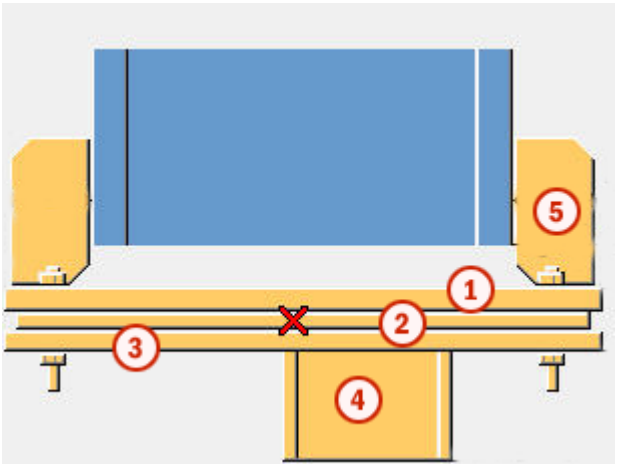
Create a column.

Selection order

1. Select the main part (column).
2. Pick a position.

   The detail is created automatically.

Part identification key

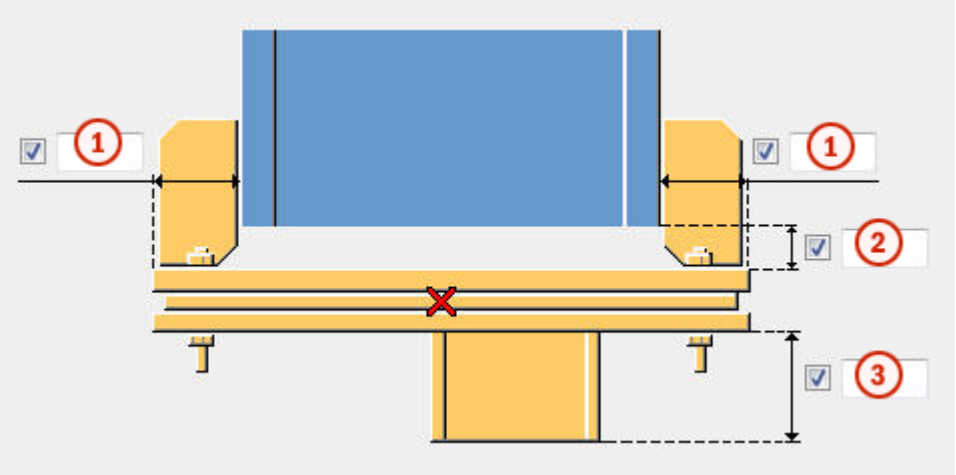


	Part
1	Base plate
2	Shim plate
3	Leveling plate
4	Shear key
5	Stiffener

Picture tab

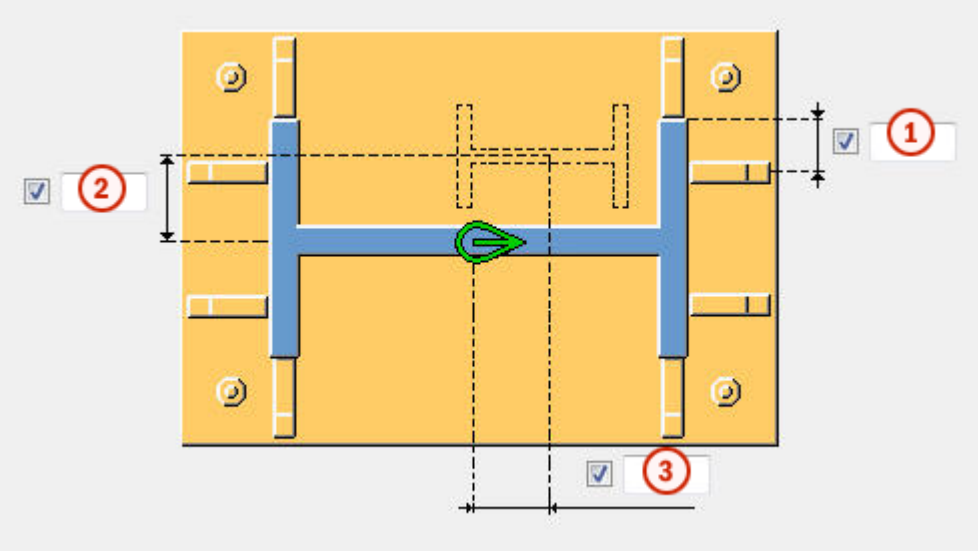
Use the **Picture** tab to control the position of the base plate and the length and position of the shear key.

Dimensions



	Description	Default
1	Distance from the flange to the edge of the base plate. Enter a negative value to make the base plate larger.	
2	Weld gap.	
3	Height of the shear key.	`Equal to the column width`

Shear key offset



	Description
1	Distance of the stiffener from the edge of the column flange.
2	Shear key vertical offset from the column center.

	Description
3	Shear key horizontal offset from the column center.

Parts tab

Use the **Parts** tab to control the dimensions of the base plate, web plates, flange plates, shear key, leveling plate, and shim plates.

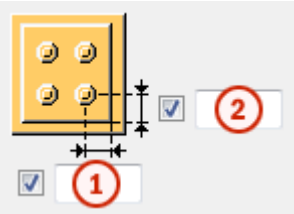
Plate

Option	Description	Default
Plate	Base plate thickness.	`BASEPLATE`
Web plates	Web plate thickness.	`10 mm`
Flange plate 4	Flange plate thickness.	`10 mm`
Leveling plate	Leveling plate thickness, width and height.	
Leveling plate hole diameter	Leveling plate hole diameter.	
Fitting plate	Shim plate thickness, width and height. Define up to three different shim plates.	
Number of fitting pl.	Number of shim plates for each thickness.	`1`
Additional beam	Shear key profile by selecting it from the profile catalog.	`HEA 300` (in Default environment)
Additional beam rotation	Select the shear key rotation type and define the rotation angle.	
Horizontal plates	Horizontal plate thickness.	
L profile, flange	Profile for L profile, flange by selecting it from the profile catalog.	
L profile, web	Profile for L profile, web by selecting it from the profile catalog.	

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

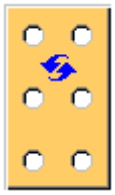
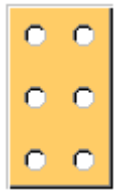
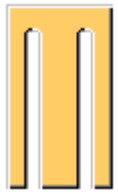
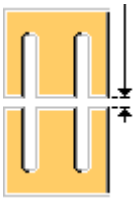
Bolt edge distances in shim plate

Define the bolt edge distances for shim plates. If you do not enter any distance, the shim plates are of the same size as the base plate.



	Description	Default
1	Horizontal bolt edge distance in the shim plate.	30 mm
2	Vertical bolt edge distance in the shim plate.	30 mm

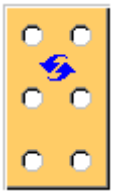
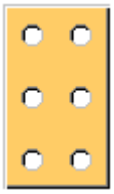
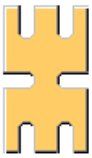
Shim plate shape

Option	Description
	Default Holes are based on the bolt group of the connection. AutoDefaults can change this option.
	Holes are based on the bolt group of the connection.
	Finger shim plate with horizontal slots. The plate can be installed from the right or the left side of the connection.
	Finger shim plate with vertical slots. The plate can be installed from the top of the connection.
	Two separate finger shim plates with horizontal slots.
	Two separate finger shim plates with vertical slots.

Tolerance for slots

Define the tolerance for the slots in finger shim plates. The width of the slot is the bolt diameter + the tolerance.

Base plate mounting grooves

Option	Description
	Default Mounting grooves are not created. AutoDefaults can change this option.
	Mounting grooves are not created.
	Mounting grooves are created horizontally.
	Mounting grooves are created vertically for the first and last row of the bolts. For other rows of bolts, the mounting grooves are created horizontally.

Tolerance for mounting grooves

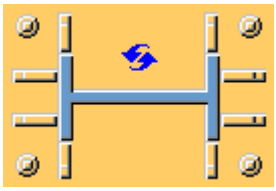
Define the tolerance for the mounting grooves in the base plates. The width of the groove is the bolt diameter + the tolerance. If you do not enter a value, the bolt tolerance value is used.

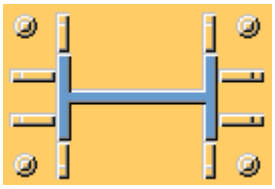
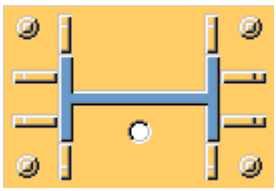
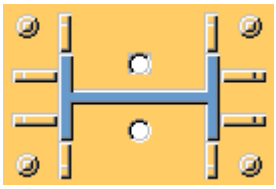
Parameters tab

Use the **Parameters** tab to control the component and the grout hole.

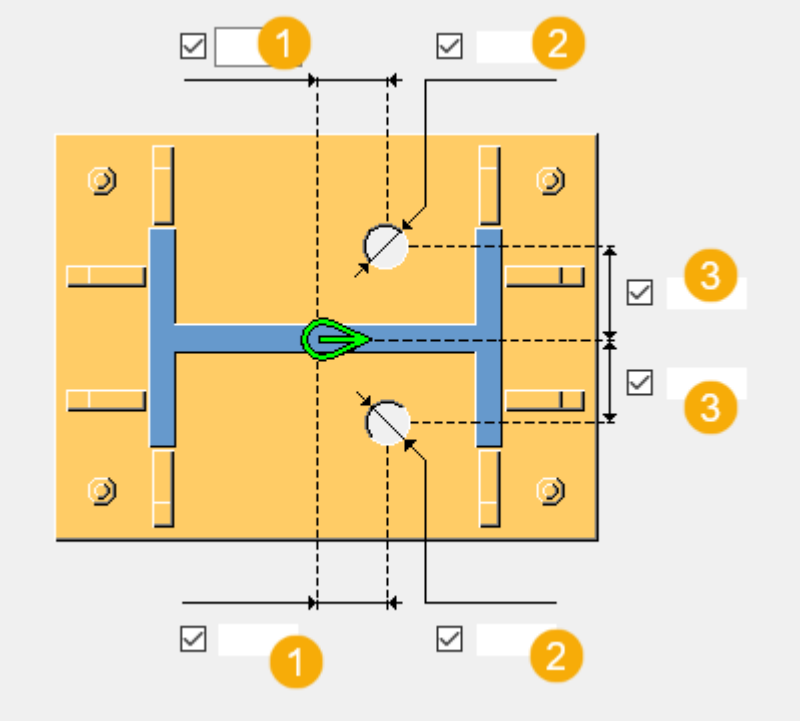
Grout hole

Define whether a grout hole is created in the base plate. The hole is also created in the leveling plate and shim plates, if they exist in the detail.

Option	Description
	Default Grout hole is not created. AutoDefaults can change this option.

Option	Description
	Grout hole is not created.
	Grout hole is created.
	Two grout holes are created.

Grout hole dimensions



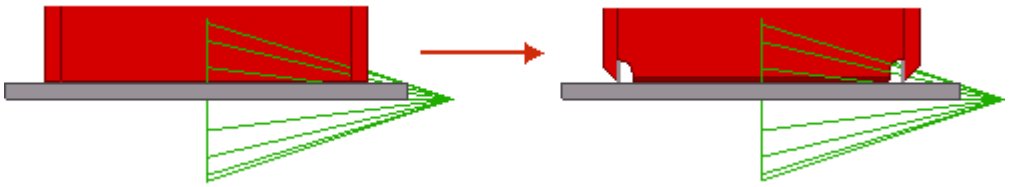
	Description
1	Location of the grout hole from the center of the column in the direction of the web.
2	Grout hole diameter.

	Description
3	Location of the grout hole from the center of the column in the direction of the flange.

Using additional components

You can use additional system or custom components to modify the column end or the base plate. For example, you can create special backing plates, weld preparations, and weld access holes for the column end.

If you use additional system or custom components, you need to manage the column end or the base plate properties in the additional component in question. When using several components, there may be multiple welds and cuts.

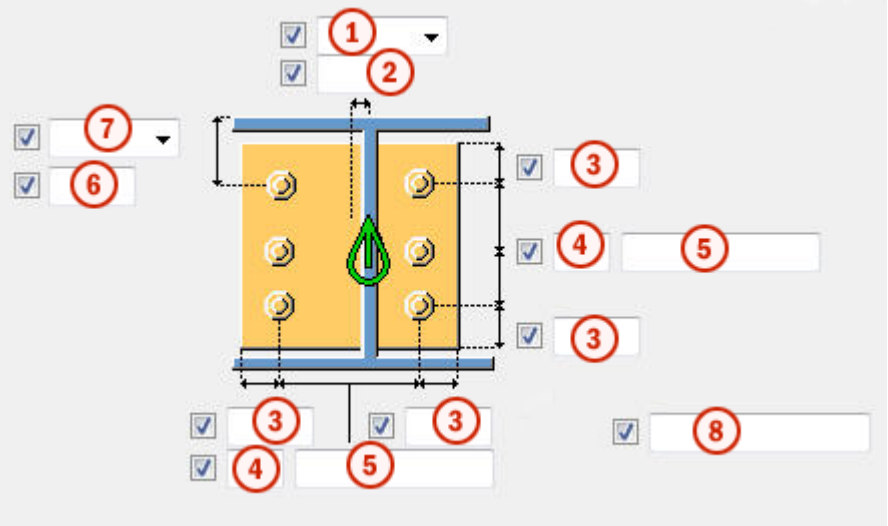


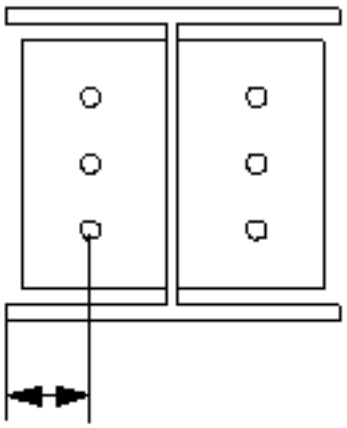
Optioon	Description
Component	Define a system or a custom component by selecting it from the **Applications & components** catalog.
Attributes	Enter the name of the attribute file for the selected component.
Input	Define to which parts the selected component is applied. • **Default** is same as **Base + Column**. • **Column** sets the column as the main part. Use this option for details. • **Column + Base** sets the column as the main part and the base plate as the secondary part. • **Base + Column** sets the base plate as the main part and the column as the secondary part. • **Base** sets the base plate as the main part.

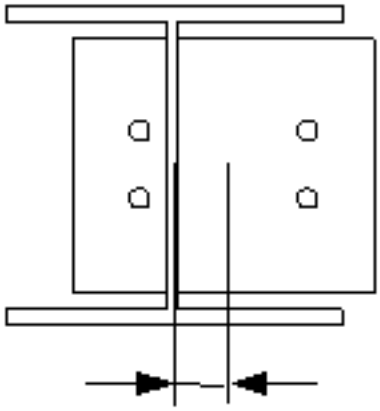
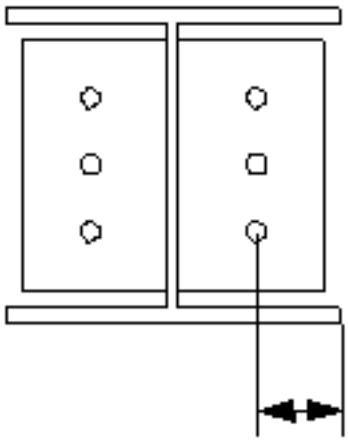
Bolts

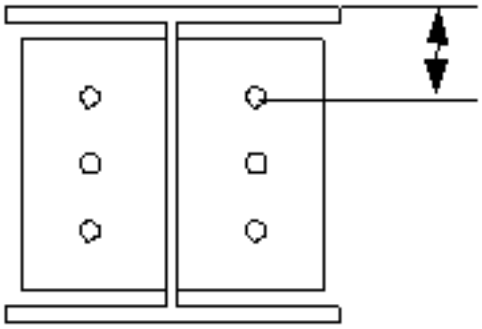
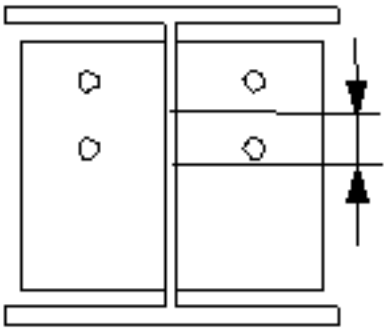
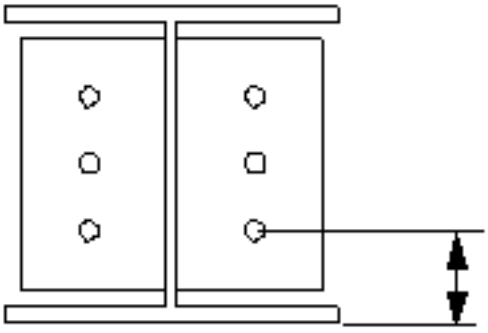
Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt. 

	Description
	• **Middle**: From the center line of the secondary part to the center line of the bolts.  • **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Dimension for vertical bolt group position.

	Description
7	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

	Description
8	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Bolt basic properties

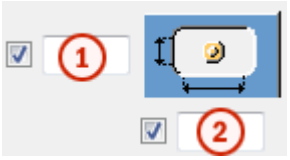
Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Slotted holes

You can define slotted, oversized, or tapped holes.



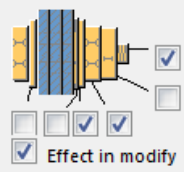
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.

Option	Description	Default
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

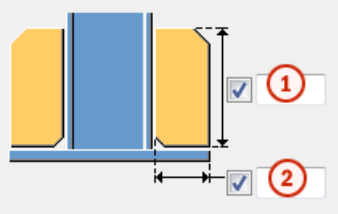
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Stiffeners

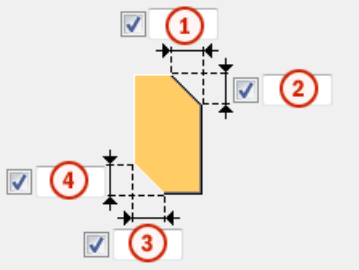
Use the **Stiffeners** tab to control the web plate stiffeners and the flange plate stiffeners.

Web plate stiffener dimensions



	Description	Default
1	Height of the web plate stiffener.	200 mm
2	Bottom width of the web plate stiffener.	100 mm

Web plate stiffener chamfer dimensions



	Description
1	Top horizontal chamfer dimension.
2	Top vertical chamfer dimension.
3	Bottom horizontal chamfer dimension.
4	Bottom vertical chamfer dimension.

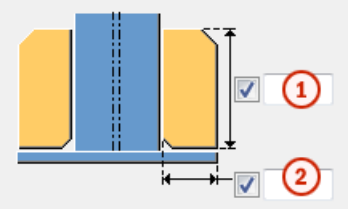
Web plate stiffener position

You can create stiffeners to different positions using the web plate stiffener position options.

Option	Description
	Default Type 2 AutoDefaults can change this option.
	Type 1
	Type 2

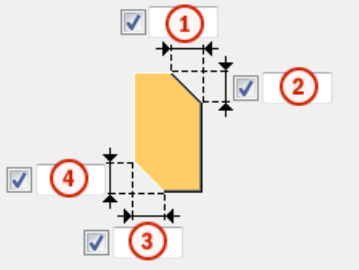
Option	Description
	Type 3
	Type 4
	Type 5
	Type 6
	Type 7

Flange plate stiffener dimensions



	Description	Default
1	Height of the flange plate stiffener.	`200 mm`
2	Bottom width of the flange plate stiffener.	`100 mm`

Flange plate stiffener chamfer dimensions



	Description
1	Top horizontal chamfer dimension.

	Description
2	The top vertical chamfer dimension.
3	Bottom horizontal chamfer dimension.
4	Bottom vertical chamfer dimension.

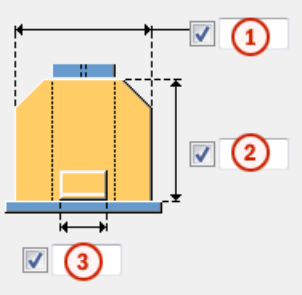
Flange plate stiffener position

You can create stiffeners to different positions using the flange plate stiffener position options.

Option	Description
	Default Type 2 AutoDefaults can change this option.
	Type 1
	Type 2
	Type 3
	Type 4

Stiffener dimensions

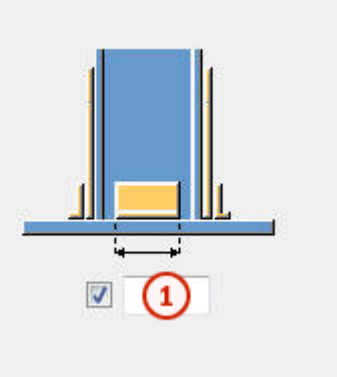
If you have created the stiffeners using other than the default stiffener position options, you can separately define the plate dimensions for different positions.



	Description
1	Width of the stiffener.
2	Height of the stiffener.
3	Width of the stiffener.

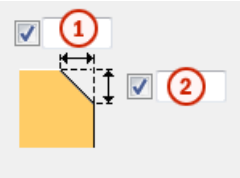
Stiffener plate width

If you have selected type 5 as the web plate stiffener position or type 4 as the flange plate stiffener position, you can define the stiffener width.



	Description	Default
1	Width of the stiffener.	200 mm

Stiffener chamfer dimensions



	Description
1	Horizontal chamfer dimension.
2	Vertical chamfer dimension.

Anchor rods tab

Use the **Anchor rods** tab to control the creation of different types of anchor rods.

Anchor rod dimensions

Option	Description
Rod profile	Anchor rod profile. You can add a comment about the part.
Nut profile	Nut profile.
Washer profile	Washer profile.
Plate washer	Plate washer thickness, width and height.

Option	Description
Cast plate	Cast plate thickness, width and height.
Grout	Grout thickness. Grouting helps you to model columns to the top of concrete parts and place the base plate correctly. It also makes it easier to dimension the detail in GA drawings. By default, no grouting is created. Select whether the grouting is created with or without slopes above or below the detail creation point. This also affects the shim plates.

Anchor rod part properties

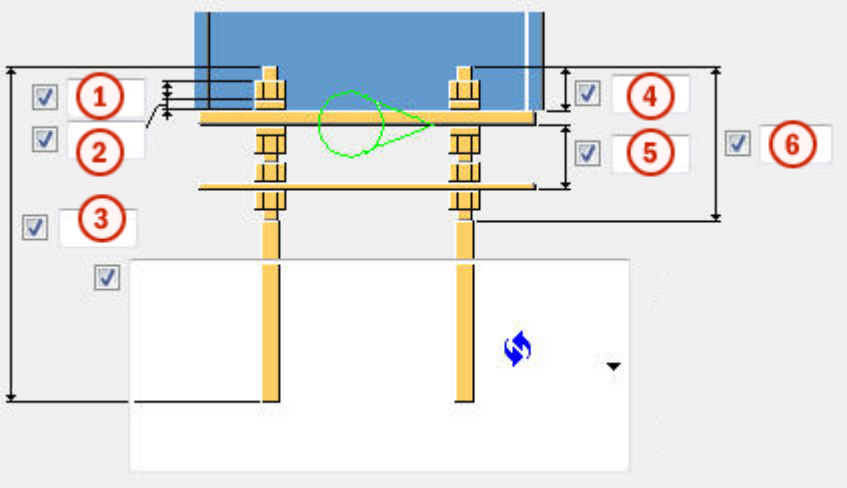
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Base plate with

Select whether to create the base plate with bolts, anchor rods, or a custom component.

By default, the base plate is created with **Bolts**.

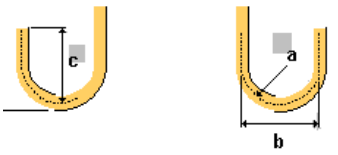
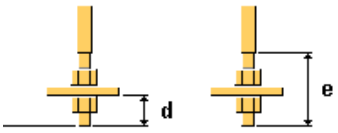
Anchor rod dimensions



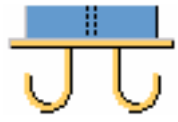
	Description	Default
1	Size or the length of the nut.	`anchor rod diameter`
2	Size or the thickness of the washer.	`half of nut size`
3	Length of the anchor rod.	`500 mm`
4	Length of the anchor rod above the base plate.	`50 mm`
5	Distance between the cast plate and the base plate.	`60 mm`
6	Length of the upper thread.	`0 mm`

Anchor rods types

Option	Description	
	Default Type 1 AutoDefaults can change this option.	
	Type 1	
	a Radius of the hook **b** Width of the hook	**a** = `2*anchor bar diameter` **b** = `1/5 of anchor bar length`

Option	Description	
	**a** Radius of the hook **b** Width of the hook **c** Height of the hook	**c** = `same as width of the hook`
	**d** Length of the anchor rod below the extra plate **e** Length of the lower thread	**d** = `2*nut size` **e** = `4*nut size plus thickness of extra plate`

Hook direction

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2
	Type 3
	Type 4

Bolting direction

NOTE You can define the bolting direction if you have created the base plate with bolts.

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1
	Bolting direction 2

Cast plate holes tolerance

Option	Description	Default
Cast plate holes tolerance	Tolerance of the cast plate holes.	`same as bolt tolerance`

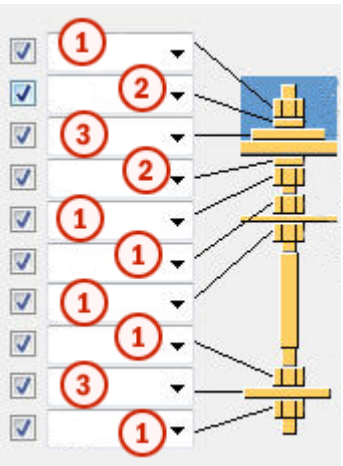
Washer hole tolerance

Option	Description
Create hole in washer	By default, a hole is not created in the washer. Tolerance of the washer hole.

Create assembly from all anchors

Define whether anchors are included in an anchor rod assembly. You can also include leveling plates into the assembly.

Create



	Description
1	Create the nut profile. You can select to create two nuts at the top of the anchor rods.
2	Create the washer profile.
3	Create the washer plate.

Anchor rod assembly

Define which parts of the anchor rod are included in the anchor rod assembly.

You can weld the washer plates above and below the base plate.

Extra plates tab

Use the **Extra plates** tab to control the placement, rotation, and type of the profiles (extra profile 1) created at the bottom of each anchor bar and the profiles (extra profile 2) that connect rows of anchor bars.

Part dimensions

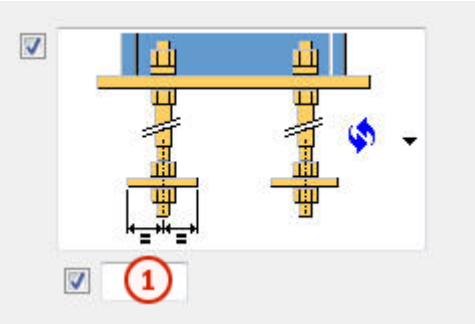
Option	Description	Default
Extra profile 1	First extra profile by selecting it from the profile catalog.	`PL10*100`
Extra profile 2	Second extra profile by selecting it from the profile catalog.	

Part properties

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

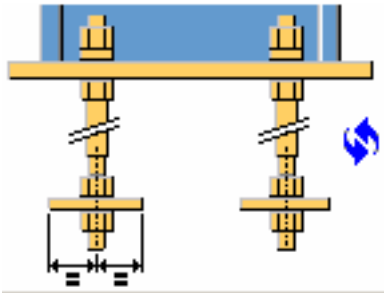
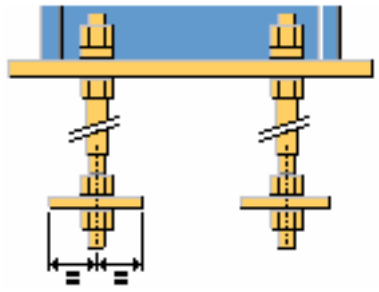
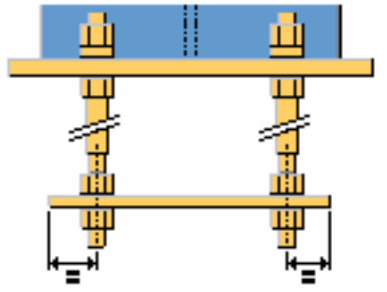
Option	Description	Default
Class	Part class number.	

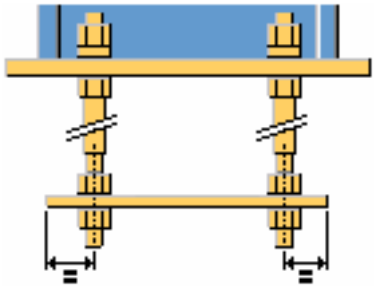
Edge distance of extra profile 1



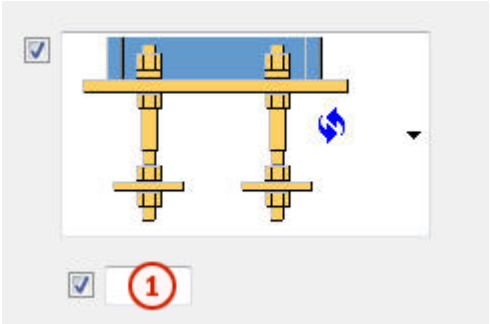
	Description	Default
1	Edge distance of extra profile 1.	50 mm

Type and direction of extra profile 1

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2

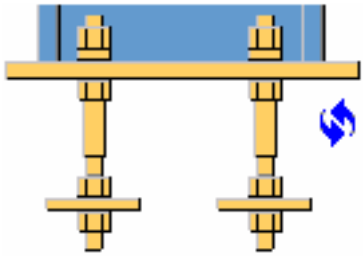
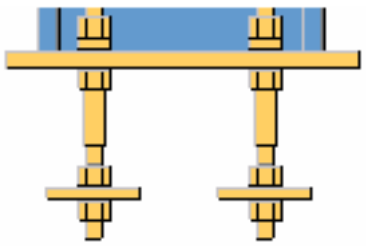
Option	Description
	Type 3

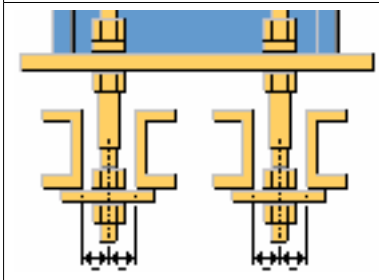
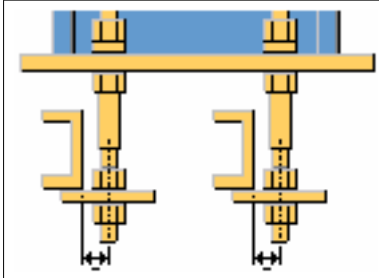
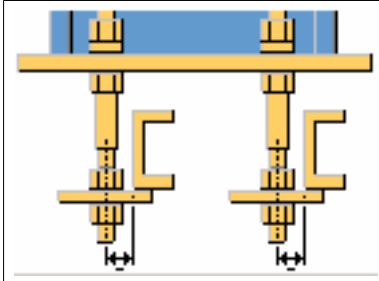
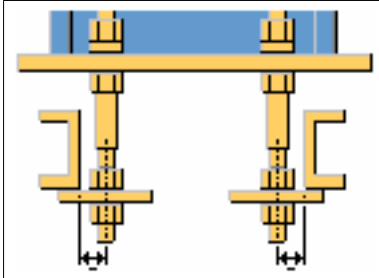
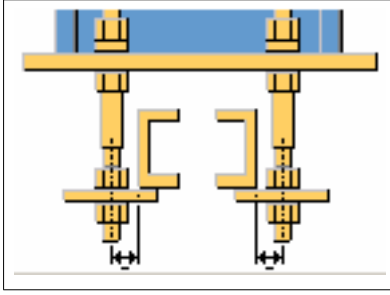
Edge distance of extra profile 2



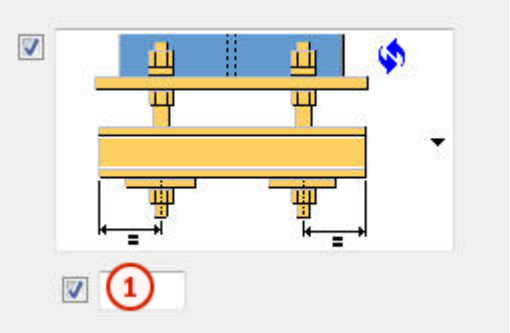
	Description	Default
1	Distance of extra profile 2 from the axis of the anchor bar.	`Half of the nut size or anchor bar diameter`

Extra profile 2 type

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1

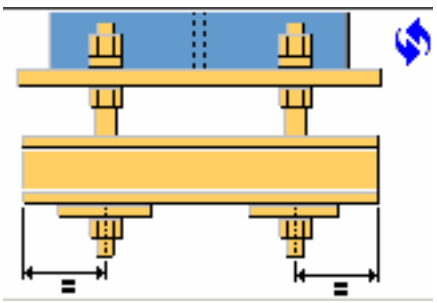
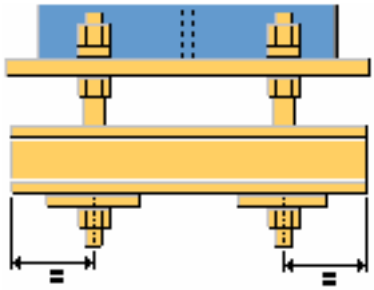
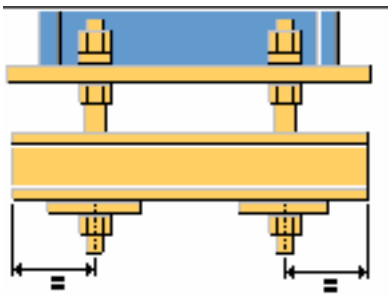
Option	Description
	Type 2
	Type 3
	Type 4
	Type 5
	Type 6

Length of extra profile 2



	Description	Default
1	Length of extra profile 2 from the axis of the anchor bar.	50 mm

Direction of extra profile 2

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2

Extra profile 1 properties

Option	Description	Default
Hole tolerance	Hole tolerance of extra profile 1.	`Same as bolt tolerance`
Circular profile height	Height of circular extra profile 1.	
Profile rotation	Profile rotation of extra profile 1.	`Front`

Extra profile 2 rotation

Option	Description	Default
Extra profile 2 rotation	Profile rotation of extra profile 2.	`Front`

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Web stiffened base plate (1016)

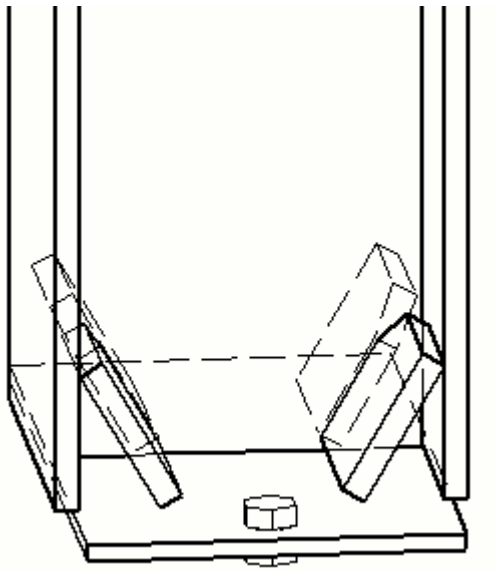
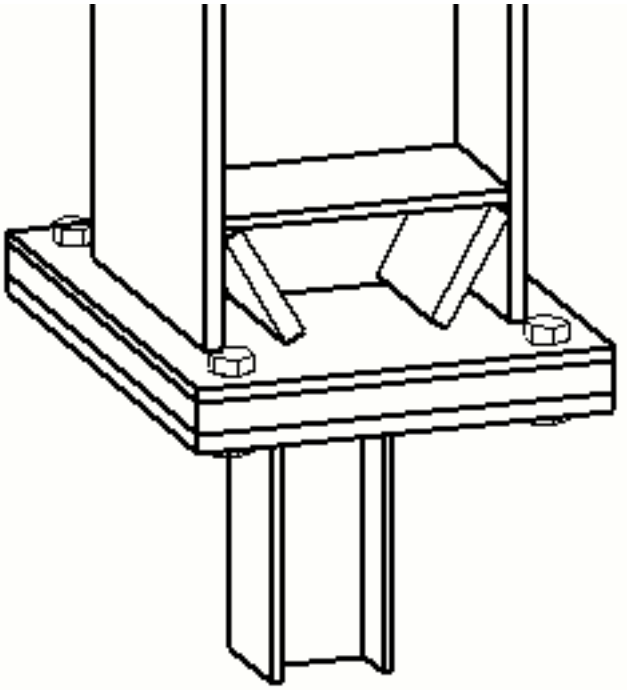
Web stiffened base plate (1016) creates a base plate with vertical, horizontal, and sloping web stiffeners.

Objects created

- Base plate
- Stiffeners
- Shim plates (optional)
- Leveling plate (optional)
- Shear key (optional)
- Extra plates connecting the anchor rods

- Anchor rods
- Bolts
- Welds
- Additional component (optional)

Use for

Situation	Description
	Web stiffened base plate
	Web stiffened base plate with a leveling plate, a shim plate, horizontal stiffener and a shear key

Before you start

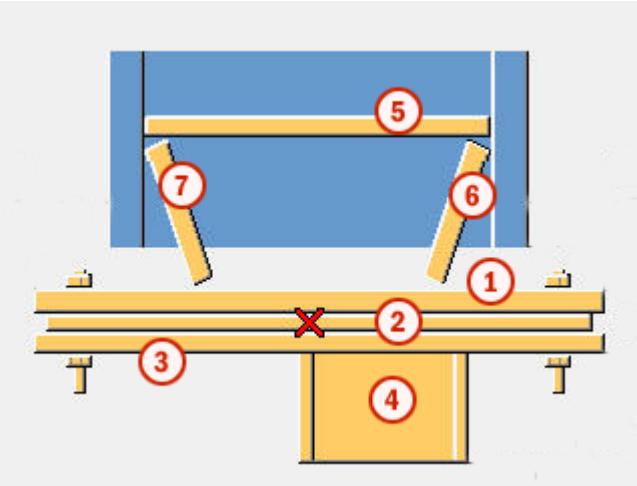
Create a column or a beam.

Selection order

1. Select the main part (column or beam).
2. Pick a position.

   The detail is created automatically.

Part identification key

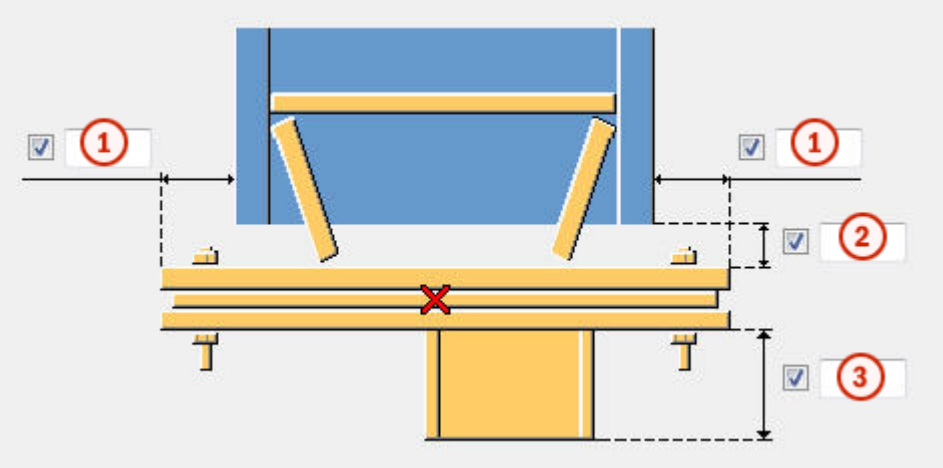


	Part
1	Base plate
2	Shim plate
3	Leveling plate
4	Shear key
5	Upper horizontal stiffener
6	Upper flange stiffener
7	Lower flange stiffener

Picture tab

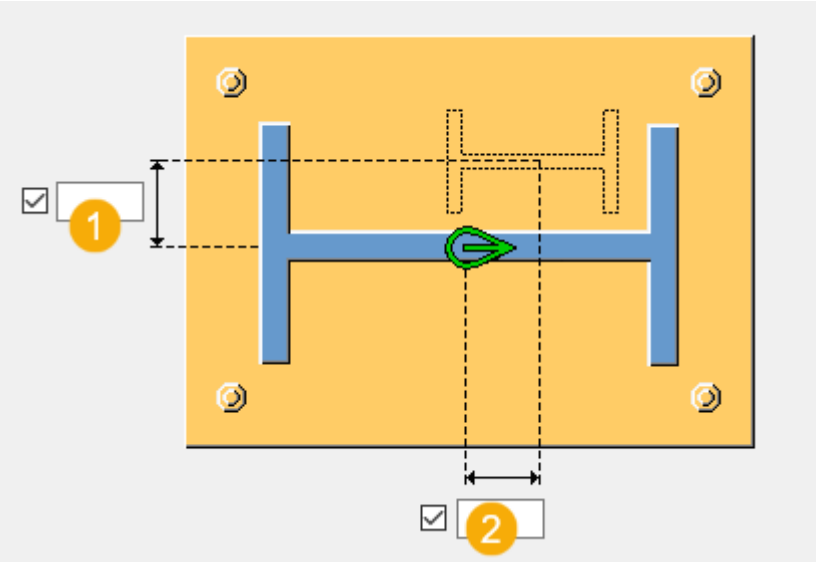
Use the **Picture** tab to control the position of the base plate and the shear key offset.

Dimensions



	Description
1	Distance from the flange to the edge of the base plate.
2	Weld gap.
3	Height of the shear key.

Shear key offset



	Description
1	Shear key vertical offset from the main part center.
2	Shear key horizontal offset from the main part center.

Parts tab

Use the **Parts** tab to control the dimensions of the base plate, upper and lower flange stiffener, shear key, leveling plate, upper horizontal stiffener, and shim plates.

Plate

	Description	Default
Plate	Base plate thickness. The dimensions on the **Picture** and **Bolts** tab determine the base plate width and length.	
Upper fl. stiffener	Upper flange stiffener thickness, width and height.	Thickness = `thickness of the main part flange` Height = `determined by the size of the main part` Width = `inner side flange distance`
Lower fl. stiffener	Lower flange stiffener thickness, width and height.	Thickness = `thickness of the main part flange` Height = `determined by the size of the main part` Width = `inner side flange distance`
Additional beam	Select the shear key profile from the profile catalog.	
Additional beam rotation	Select the shear key rotation type and define the rotation angle.	
Additional beam welded to	Define to which plate the shear key is welded.	
Leveling plate	Leveling plate thickness, width and height.	
Leveling plate hole diameter	Leveling plate hole diameter.	
Upper horizontal stiff	Upper horizontal stiffener thickness, width and height.	Thickness = `0 mm`

	Description	Default
Fitting plate	Shim plate thickness, width and height. Define up to three different shim plates.	
Number of fitting pl.	Number of shim plates for each thickness.	1

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

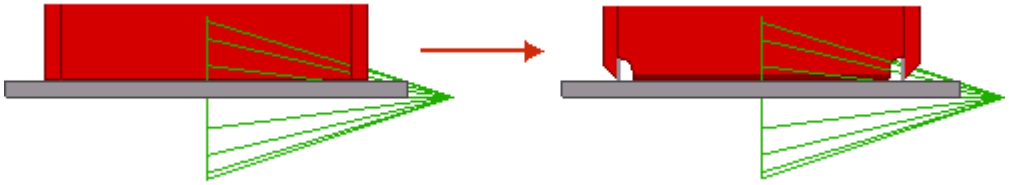
Parameters tab

Use the **Parameters** tab to control the component used inside **Web stiffened base plate (1016)**.

Using additional components

You can use additional system or custom components to modify the column end or the base plate. For example, you can create special backing plates, weld preparations, and weld access holes for the column end.

If you use additional system or custom components, you need to manage the column end or the base plate properties in the additional component in question. When using several components, there may be multiple welds and cuts.

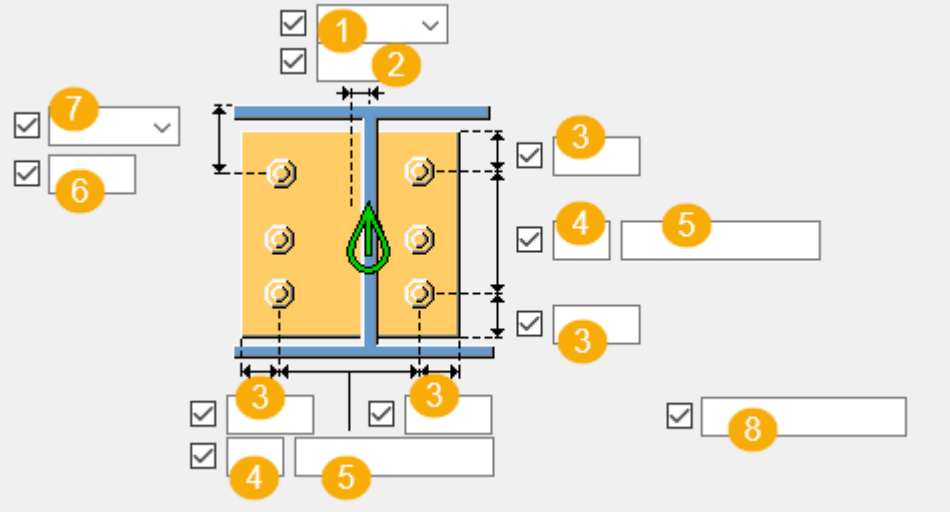


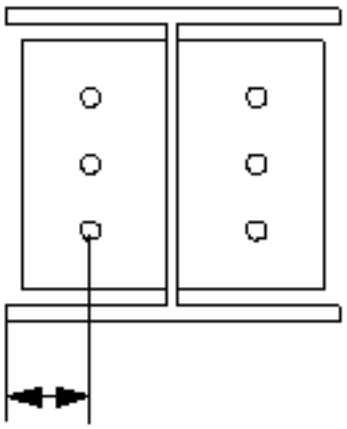
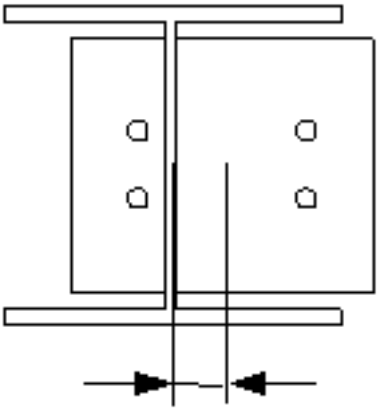
Part	Description
Component	Define a system or a custom component by selecting it from the component catalog.
Attributes	Enter the name of the attribute file for the selected component.
Input	Define to which parts the selected component is applied. • **Default** is same as **Base + Column**. • **Column** sets the column as the main part. Use this option for details. • **Column + Base** sets the column as the main part and the base plate as the secondary part. • **Base + Column** sets the base plate as the main part and the column as the secondary part. • **Base** set the base plate as the main part.

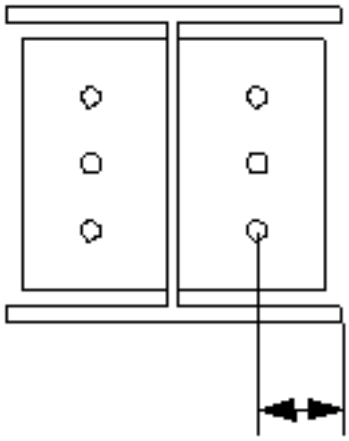
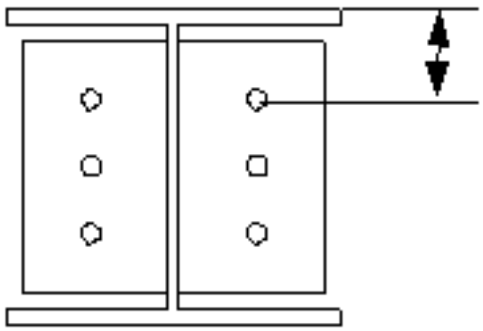
Bolts tab

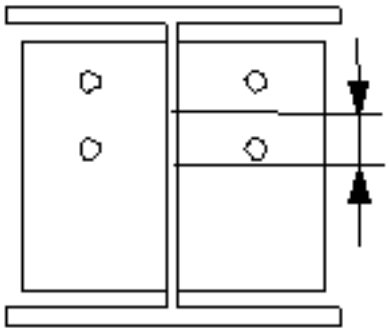
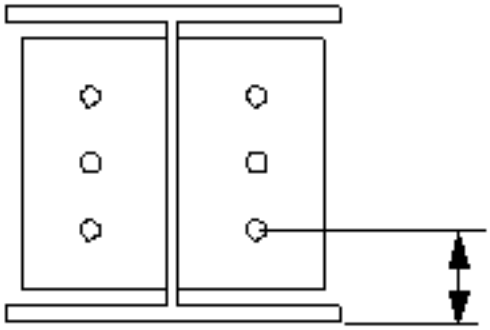
Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt.  • **Middle**: From the center line of the secondary part to the center line of the bolts. 

	Description
	• **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Dimension for vertical bolt group position.
7	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
8	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

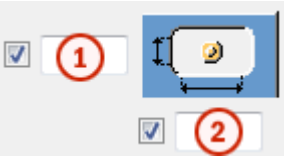
Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Slotted holes

You can define slotted, oversized, or tapped holes.

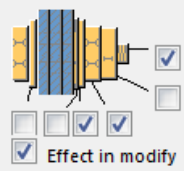


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

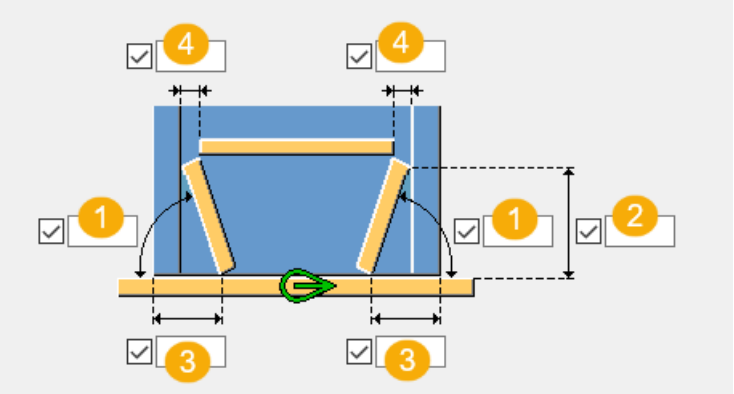
Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Stiffeners tab

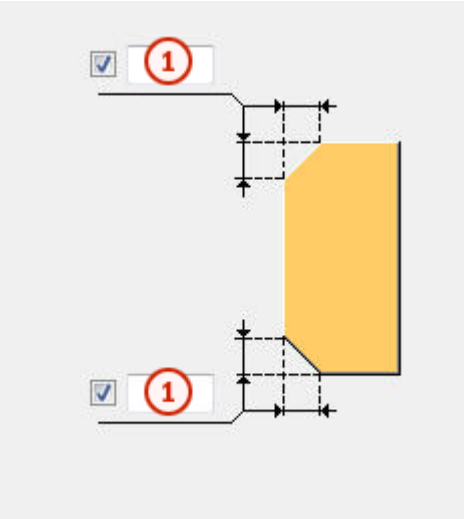
Use the **Stiffeners** tab to control the stiffener angle, position and selection.

Stiffener angle and position



	Description	Default
1	Stiffener angle.	`60 degrees`
2	Height of the stiffener.	
3	Distance of the stiffener from the column edge.	
4	Gap between the upper horizontal stiffener and the main part flange.	

Stiffener plate dimension



	Description
1	Size of the chamfer.

Stiffener selection 1

Option	Description
	Default Stiffeners are sloped. AutoDefaults can change this option.
	Stiffeners are sloped.
	Stiffeners are vertical.

Stiffener selection 2

Option	Description
	Default Stiffeners are sloped. AutoDefaults can change this option.
	Stiffeners are sloped.
	Flanges are cut and stiffeners are created. If you have selected the vertical stiffeners, you cannot create sloped stiffeners or cut flanges.

Anchor rods tab

Use the **Anchor rods** tab to control the creation of different types of anchor rods.

Anchor rod dimensions

Option	Description
Rod profile	Anchor rod profile.
Nut profile	Nut profile.
Washer profile	Washer profile.
Plate washer	Plate washer thickness, width and height.
Cast plate	Cast plate thickness, width and height.

Option	Description
Grout	Grout thickness. Grouting helps you to model columns to the top of concrete parts and place the base plate correctly. It also makes it easier to dimension the detail in GA drawings. By default, no grouting is created. Select whether the grouting is created with or without slopes above or below the detail creation point. This also affects the shim plates.

Anchor rod part properties

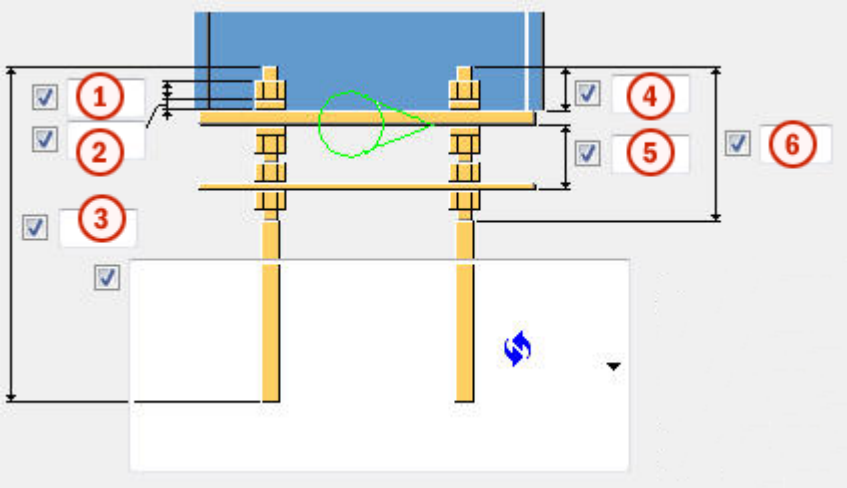
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Comment	Add a comment about the part.	

Base plate with

Select whether to create the base plate with bolts, anchor rods, or a custom component.

By default, the base plate is created with **Bolts**.

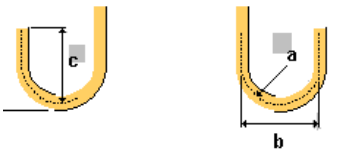
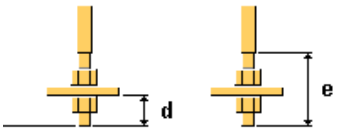
Anchor rod dimensions



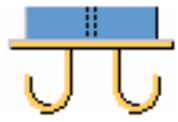
	Description	Default
1	Size or the length of the nut.	`anchor rod diameter`
2	Size or the thickness of the washer.	`half of nut size`
3	Length of the anchor rod.	`500 mm`
4	Length of the anchor rod above the base plate.	`50 mm`
5	Distance between the cast plate and the base plate.	`60 mm`
6	Length of the upper thread.	`0 mm`

Anchor rods types

Option	Description	
	Default Type 1 AutoDefaults can change this option.	
	Type 1	
	a Radius of the hook **b** Width of the hook	**a** = `2*anchor bar diameter` **b** = `1/5 of anchor bar length`

Option	Description	
	**a** Radius of the hook **b** Width of the hook **c** Height of the hook	**c** = `same as width of the hook`
	**d** Length of the anchor rod below the extra plate **e** Length of the lower thread	**d** = `2*nut size` **e** = `4*nut size plus thickness of extra plate`

Hook direction

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2
	Type 3
	Type 4

Bolting direction

NOTE You can define the bolting direction if you have created the base plate with bolts.

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1
	Bolting direction 2

Cast plate holes tolerance

Option	Description	Default
Cast plate holes tolerance	Tolerance of the cast plate holes.	`same as bolt tolerance`

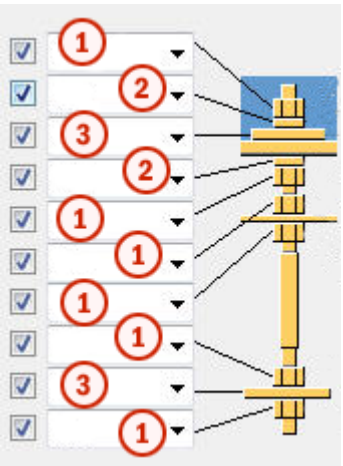
Washer hole tolerance

Option	Description
Create hole in washer	By default, a hole is not created in the washer. Tolerance of the washer hole.

Create assembly from all anchors

Define whether anchors are included in an anchor rod assembly. You can also include leveling plates into the assembly.

Create



	Description
1	Create the nut profile.
2	Create the washer profile.
3	Create the washer plate.

Anchor rod assembly

Define which parts of the anchor rod are included in the anchor rod assembly.

You can weld the washer plates above and below the base plate.

Extra plates tab

Use the **Extra plates** tab to control the placement, rotation, and type of the profiles (extra profile 1) created at the bottom of each anchor bar and the profiles (extra profile 2) that connect rows of anchor bars.

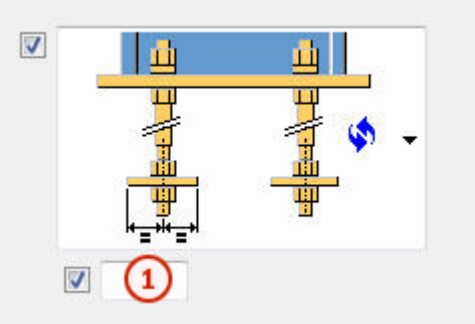
Part dimensions

Option	Description	Default
Extra profile 1	First extra profile by selecting it from the profile catalog.	`PL10*100`
Extra profile 2	Second extra profile by selecting it from the profile catalog.	

Part properties

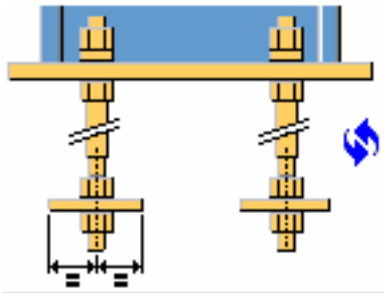
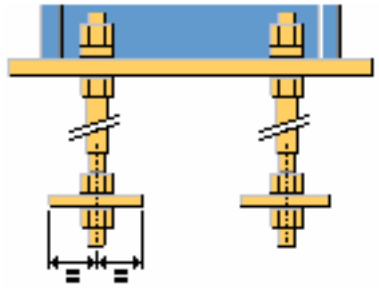
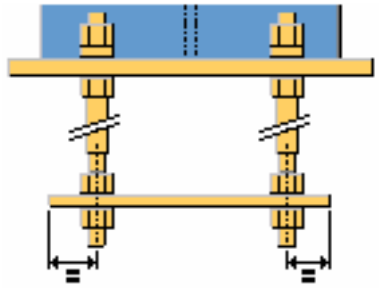
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

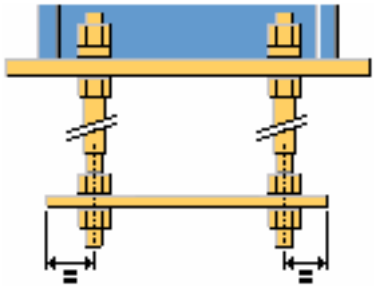
Edge distance of extra profile 1



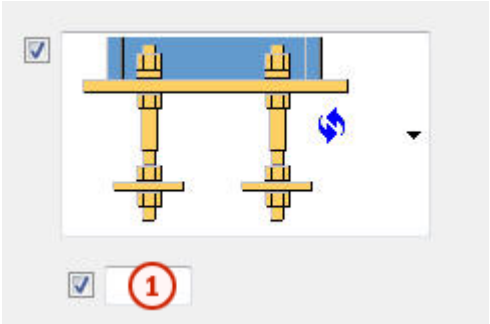
	Description	Default
1	Edge distance of extra profile 1.	50 mm

Type and direction of extra profile 1

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2

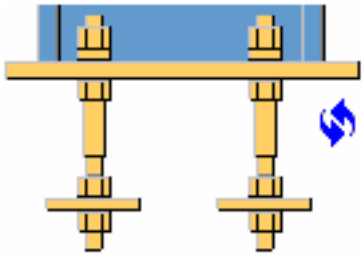
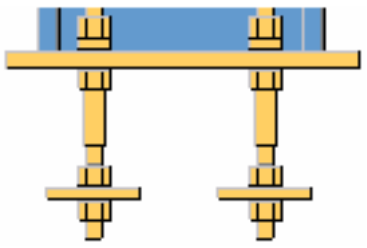
Option	Description
	Type 3

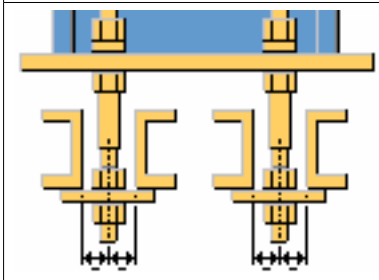
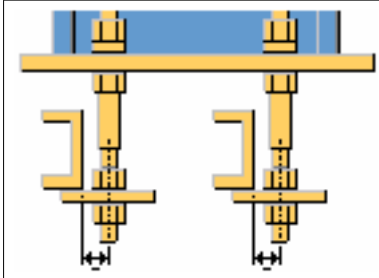
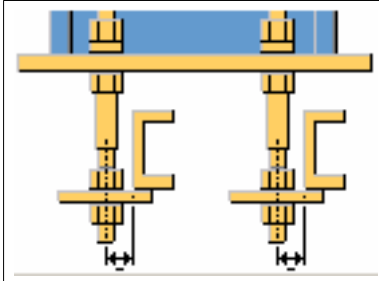
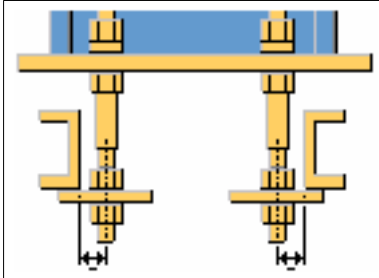
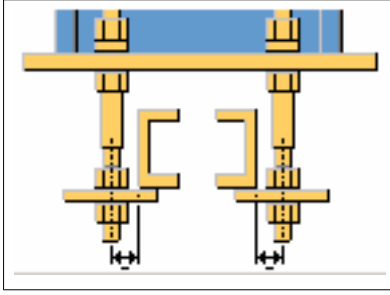
Edge distance of extra profile 2



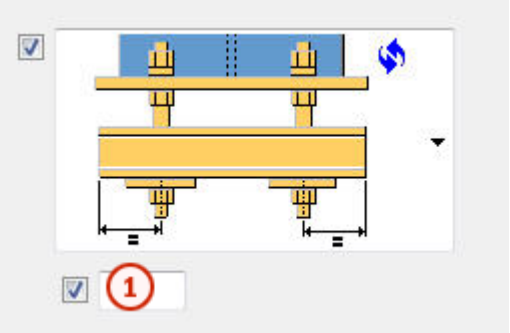
	Description	Default
1	Distance of extra profile 2 from the axis of the anchor bar.	`Half of the nut size or anchor bar diameter`

Extra profile 2 type

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1

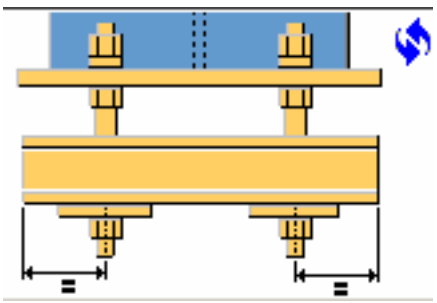
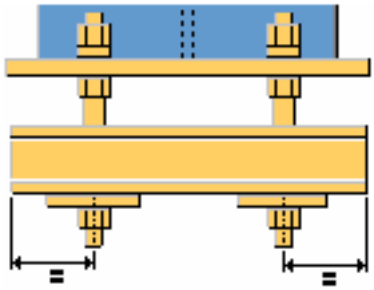
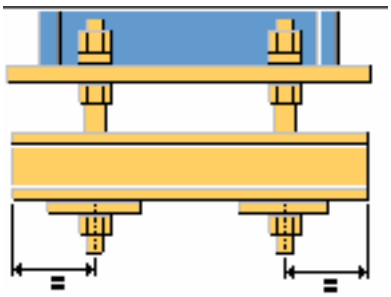
Option	Description
	Type 2
	Type 3
	Type 4
	Type 5
	Type 6

Length of extra profile 2



	Description	Default
1	Length of extra profile 2 from the axis of the anchor bar.	50 mm

Direction of extra profile2

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2

Extra profile 1 properties

Option	Description	Default
Hole tolerance	Hole tolerance of extra profile 1.	`Same as bolt tolerance`
Profile rotation	Profile rotation of extra profile 1.	`Front`

Extra profile 2 rotation

Option	Description	Default
Extra profile 2 rotation	Profile rotation of extra profile 2.	`Front`

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

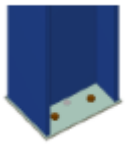
Simple base plate 2 (1031)

Simple base plate 2 (1031) creates a base plate that is connected to a column or beam end.

Objects created

- Base plate
- Stiffeners (for W profiles only)
- Shear key
- Leveling plate
- Welds

Use for

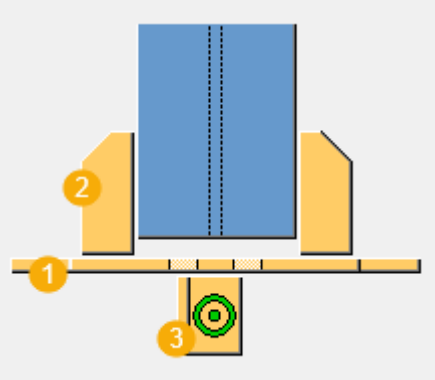
Situation	Description
	• Base plate created at column end

Selection order

1. Select the main part (column or beam).
2. Pick a position.

   The detail is created automatically.

Part identification key

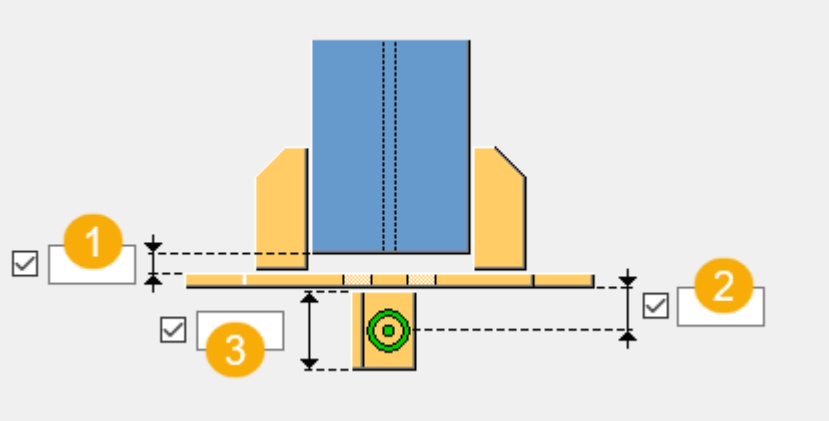


	Description
1	Base plate
2	Stiffeners
3	Shear key

Picture tab

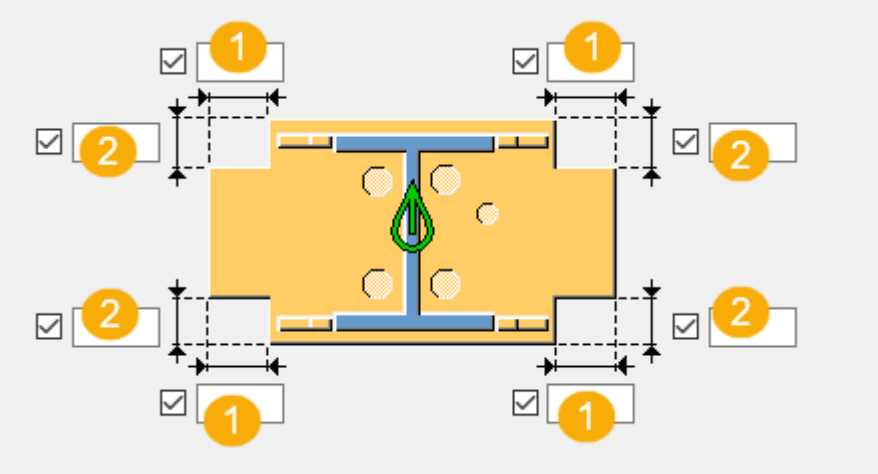
Use the **Picture** tab to define the plate position and corner cuts.

Dimensions



	Description
1	Base plate offset from the bottom edge of the column.
2	Distance from the component input point to the base plate bottom. You can offset the base plate from the point picked for creating the component.
3	Shear key height

Base plate corner cut



	Description
1	Horizontal corner cut dimension.
2	Vertical corner cut dimension.

Parts tab

Use the **Parts** tab to define the plate properties.

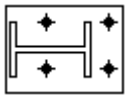
Parts

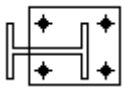
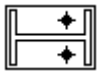
Option	Description
Base plate	Thickness, width, and height of the base plate.
Stiffeners	Thickness, width, and height of the stiffeners.
Key profile	Select the shear key profile from the profile catalog.
Leveling plate	Thickness of the leveling plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

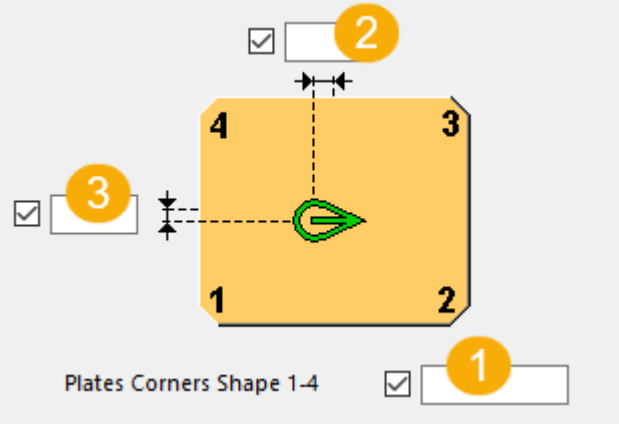
Parameters tab

Use the **Parameters** tab to define the base plate corner shape, condensation holes, shear key creation, and leveling plate extra length.

Option	Description
Plate wrap around	Define whether the base plate is automatically wrapped around the column and bolts, bolts, or column. • Around bolts and column 

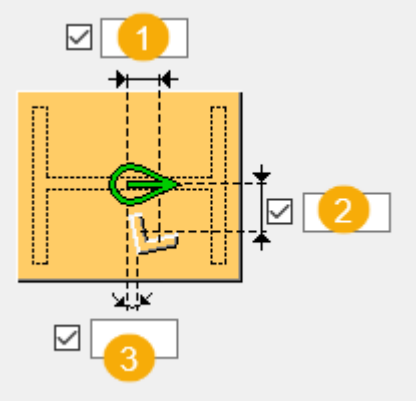
Option	Description
	• Around bolts  • Around column 
Condensation holes sides 1-4	Define where the condensation holes are located by entering the corner numbers. Semi-circular holes are placed at the bottom of columns and some circular profiles.
Condensation holes diameter	Define the diameter of the condesation holes.

Base plate corner shape



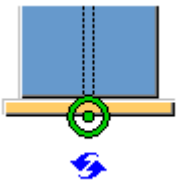
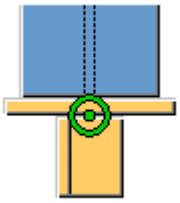
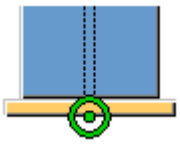
	Description
1	Define which plate corners are reshaped. You can enter the corner numbers in any order. Corners are not reshaped if you do not enter any numbers.
2	Define the horizontal offset of the plate from the bolt center line. This offset works when **Plate wrap around** is set to **Automatic**.
3	Define the vertical offset of the plate from the bolt center line. This offset works when **Plate wrap around** is set to **Automatic**.

Shear key offset and rotation

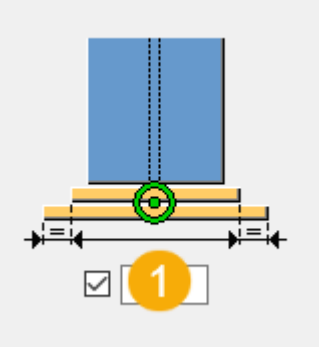


	Description
1	Define the horizontal offset of the shear key.
2	Define the vertical offset of the shear key.
3	Define the rotation of the shear key around the column center line.

Shear key creation

Option	Description
	Default Shear key is not created. AutoDefaults can change this option.
	Shear key is created.
	Shear key is not created.

Leveling plate size

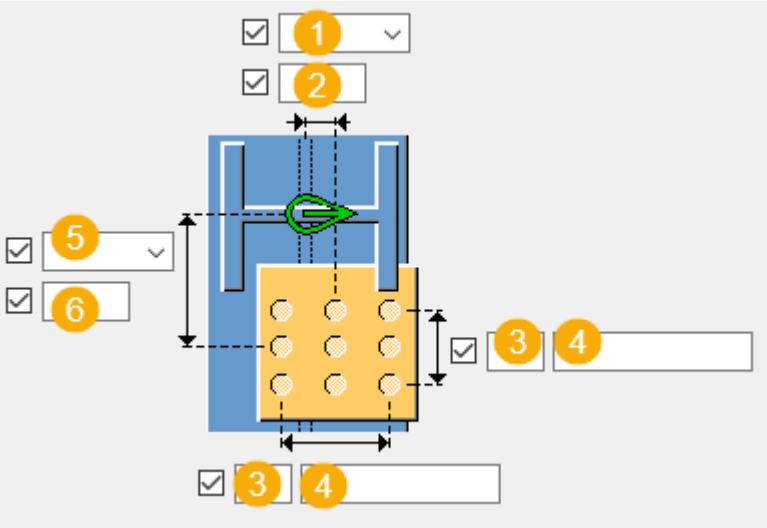


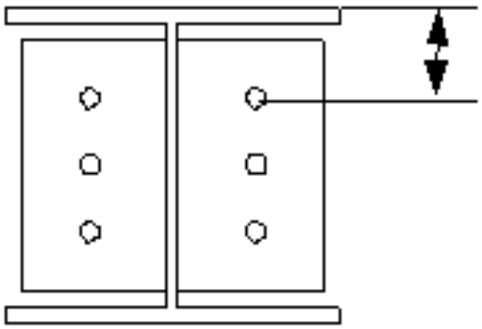
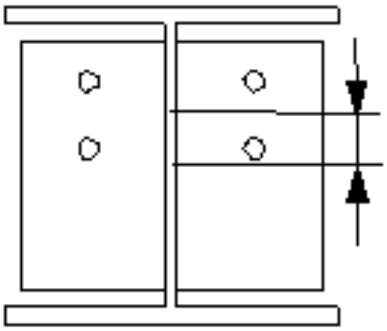
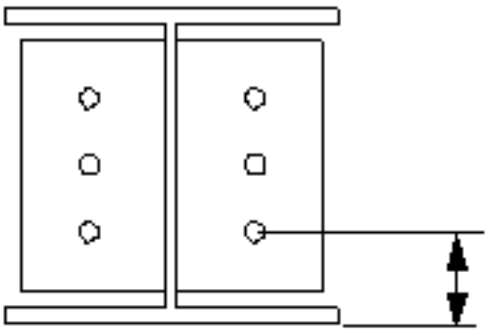
	Description
1	Define the leveling plate extra length.

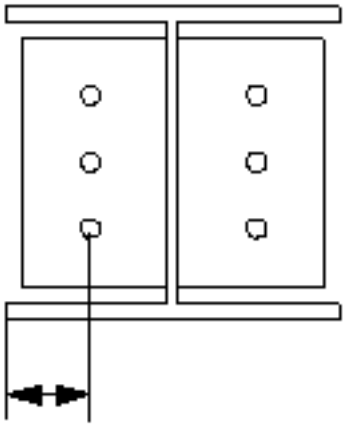
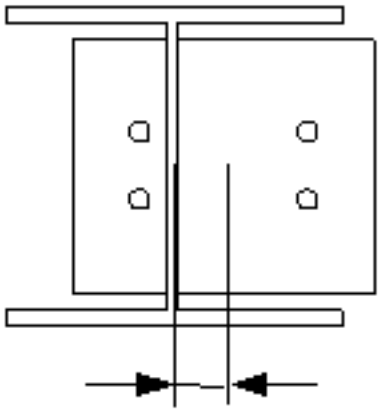
Bolts tab

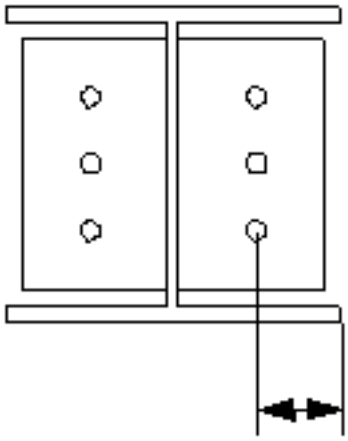
Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions

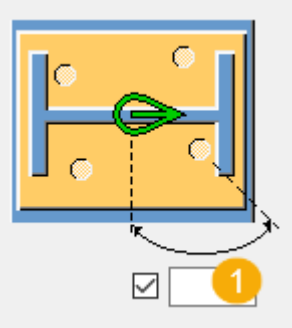


	Description
1	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
2	Dimension for vertical bolt group position.

	Description
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
5	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt.  • **Middle**: From the center line of the secondary part to the center line of the bolts. 

	Description
	• **Right**: From the right edge of the secondary part to the rightmost bolt. 
6	Dimension for horizontal bolt group position.

Bolt group rotation



	Description
1	Define the bolt group rotation around the column center line, measured from the column web.

Bolt deletion

Option	Description
Edge tolerance for deleting bolts	Define the minumum bolt edge distance for deleting bolts. The distance is measured from the bolt center to the edge of the plate.
Eliminate bolts	Define which bolts are deleted. Enter the bolt numbers from the first bolt in the first row to the last bolt in the last row, for example, `1 3 10`. Enter `A` or `a` to delete all bolts.

Bolt basic properties

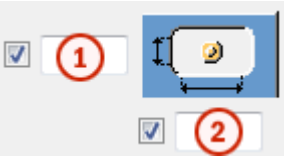
Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Slotted holes

You can define slotted, oversized, or tapped holes.



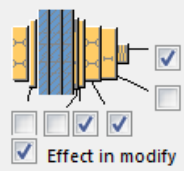
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

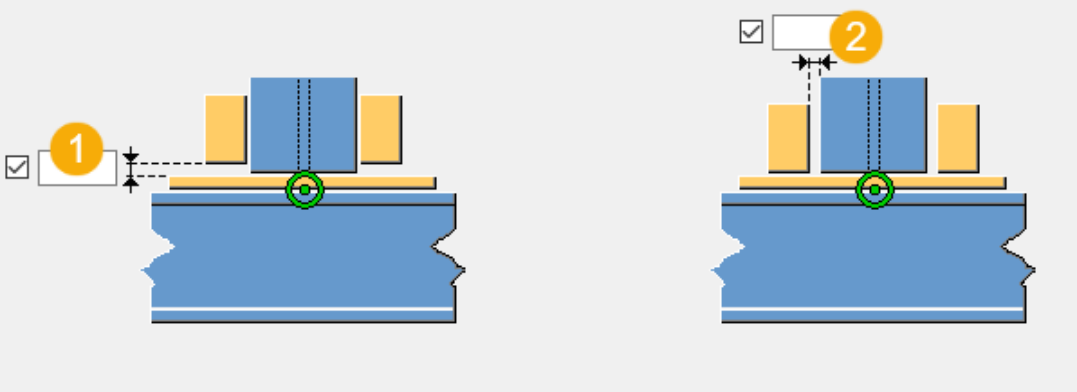
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Stiffeners tab

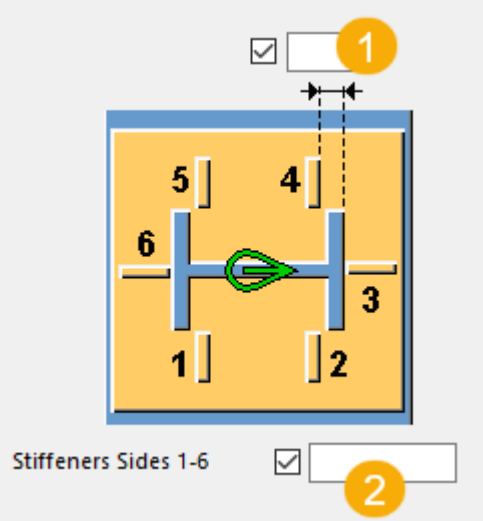
Use the **Stiffeners** tab to define the stiffener dimensions and chamfering.

Stiffener offset



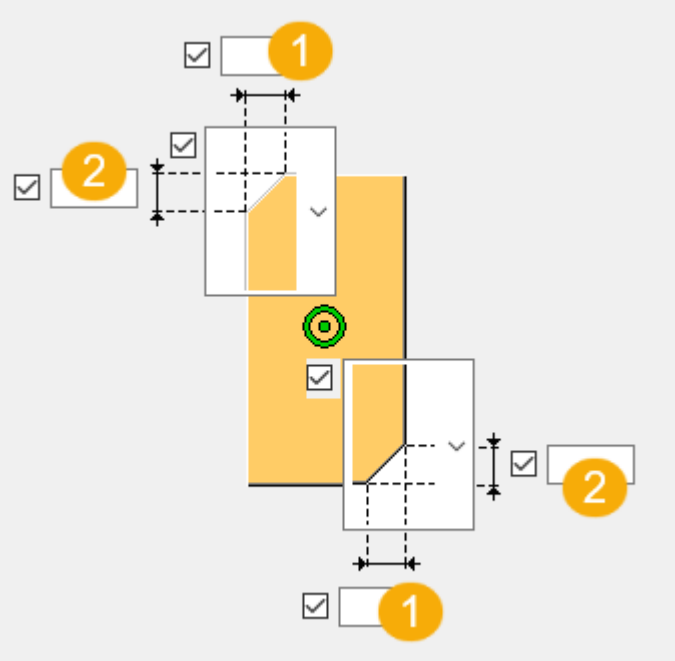
1	Stiffener offset from the base plate.
2	Stiffener offset from the column.

Stiffener positions



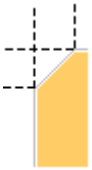
	Description
1	Stiffener position on the base plate edge.
2	Define on which sides the stiffeners are created.

Chamfer dimensions



	Description
1	Horizontal dimension of the chamfer.
2	Vertical dimension of the chamfer.

Chamfer type

Option	Description
	Default No chamfer AutoDefaults can change this option.
	Line chamfer
	Convex chamfer

Option	Description
	Concave chamfer
	No chamfer

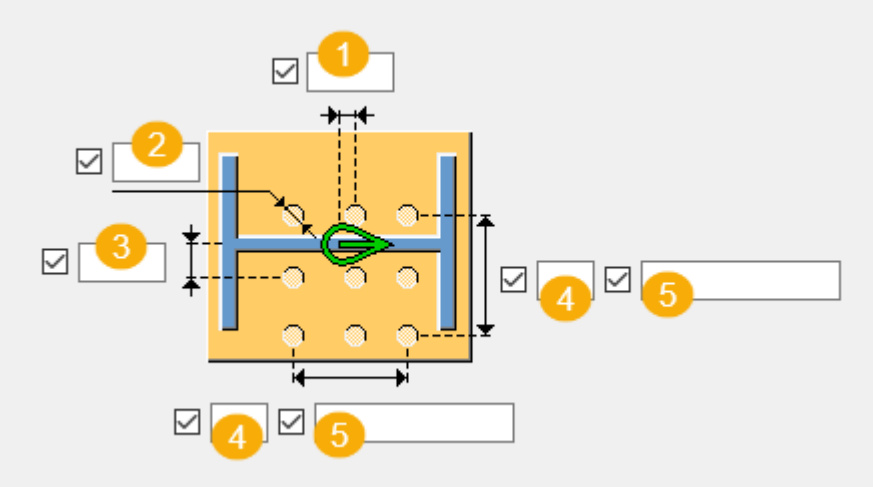
Grouting holes tab

Use the **Grouting holes** tab to define the grout hole dimensions.

Grout holes

You can define whether to delete some or all of the grout holes in **Elim holes**.

Grout hole dimensions



	Description
1	Location of the grout hole from the center of the column in the direction of the web.
2	Grout hole diameter.
3	Location of the grout hole from the center of the column in the direction of the flange.
4	Number of grout holes

	Description
5	Grout hole spacing Use a space to separate the spacing values. Enter a value for each space between holes. For example, if there are 3 bolts, enter 2 values.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

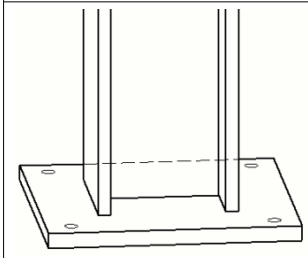
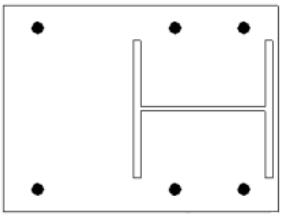
Base plate (1042)

Base plate (1042) creates a base plate that is connected to a column end.

Objects created

- Base plate
- Bolts
- Extra plates connecting the anchor rods
- Anchor rods
- Bolts
- Welds
- Additional component (optional)

Use for

Situation	Description
	Column base detail
	Column base plate at the face of the flange

Before you start

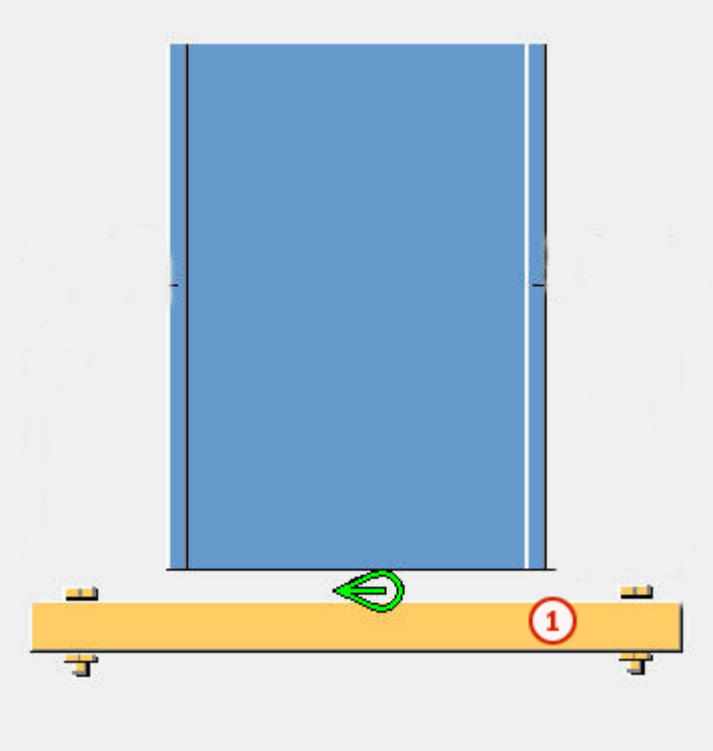
Create a column.

Selection order

1. Select the main part (column).
2. Pick a position.

   The detail is created automatically.

Part identification key

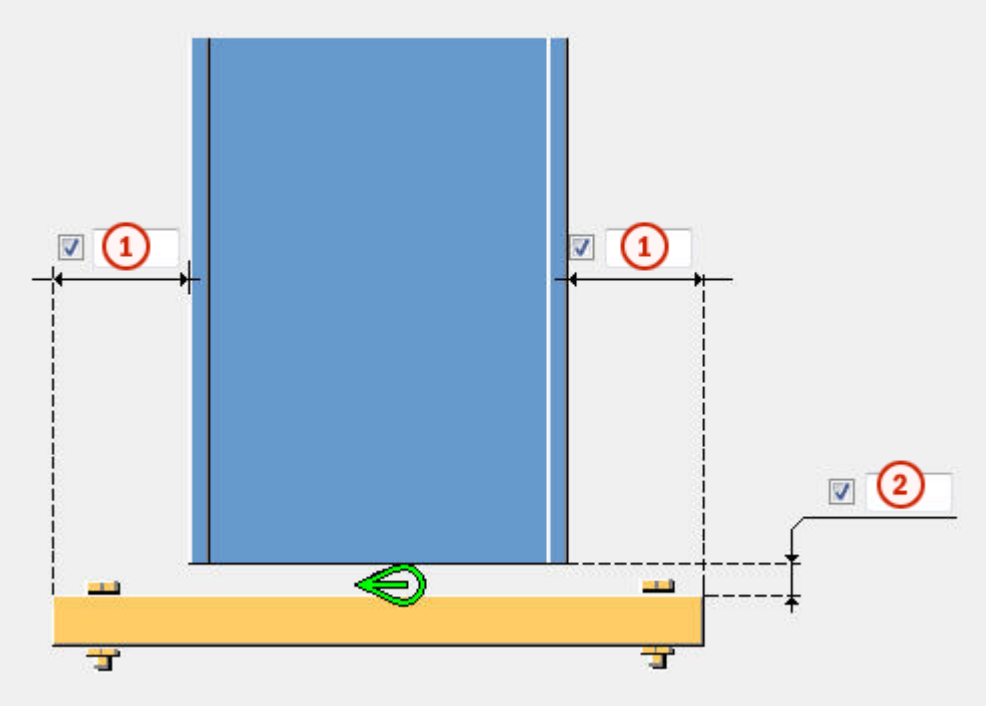


	Part
1	Base plate

Picture tab

Use the **Picture** tab to control the position of the base plate.

Dimensions



	Description
1	Distance from the flange of the main part to the edge of the base plate.
2	Weld gap.

Parts tab

Use the **Parts** tab to control the dimensions of the base plate.

Plate

Option	Description	Default
Plate	Thickness, width, and height of the base plate.	Thickness = 30 mm Width =500 mm Heigh t =500 mm

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
	assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

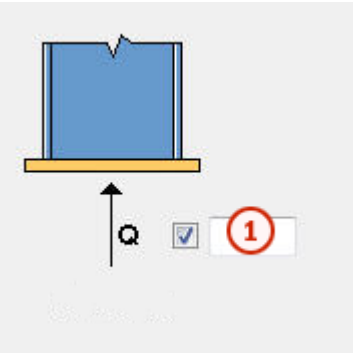
Parameters tab

Use the **Parameters** tab to control the weld dimensions, grout hole diameter and position, plate type, design check and shear force value.

Design check

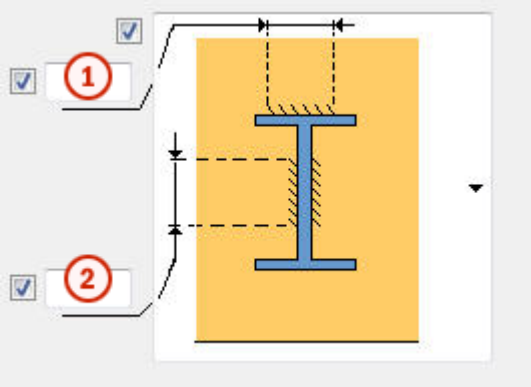
Option	Description
Design	Define whether design check is turned on or off.
Bearing	Define whether the detail and its components bear the entered shear force.
Factored Axial Dead Load (kN)	Define the factored axial dead load, if design check is turned on.
Minimum Fcu (N/mm^2)	Define the minimum Fcu, if design check is turned on.

Shear force



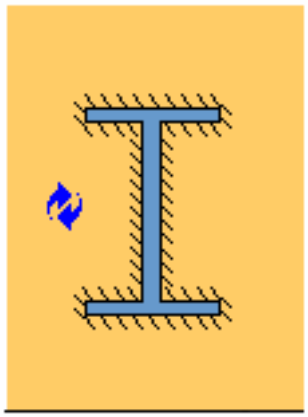
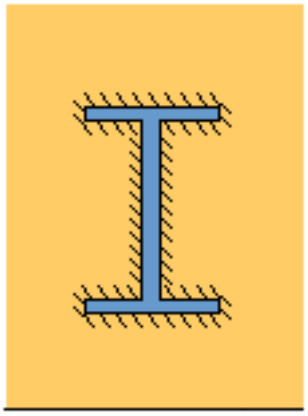
	Description
1	Shear force value. If the design check is turned on, enter a positive value. If there is no shear force, enter `0`.

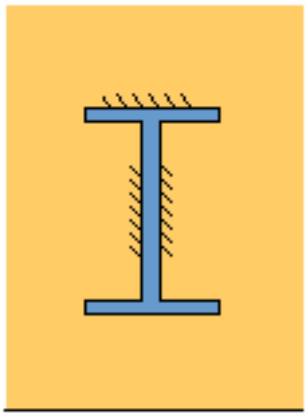
Weld dimensions



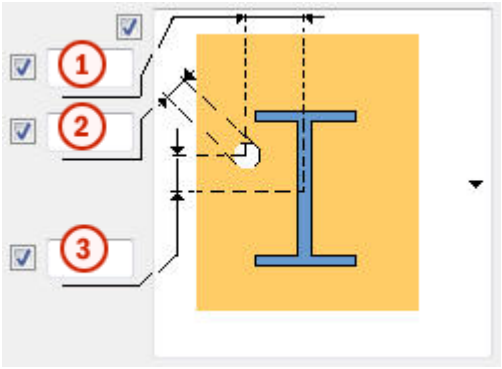
	Description
1	Weld dimension on the column flange.
2	Weld dimension on the column web.

Weld types

Option	Description
	Default Fully welded AutoDefaults can change this option.
	Fully welded

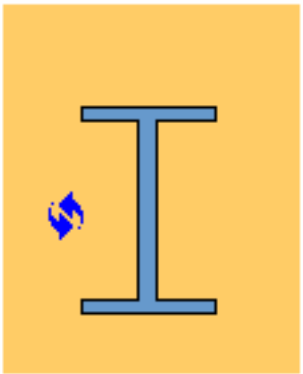
Option	Description
	Partially welded

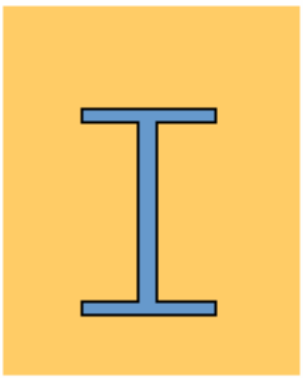
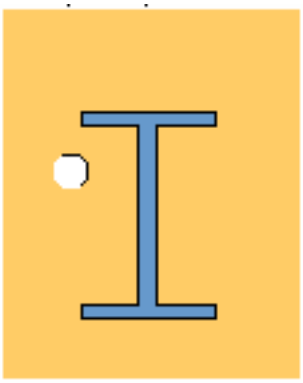
Grout hole diameter and offset



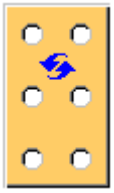
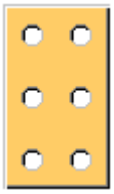
	Description
1	Grout hole horizontal offset from the column center.
2	Grout hole diameter.
3	Grout hole vertical offset from the column center.

Base plate with grout hole

Option	Description
	Default Grout hole is not created. AutoDefaults can change this option.

Option	Description
	Grout hole is not created.
	Grout hole is created.

Base plate mounting grooves

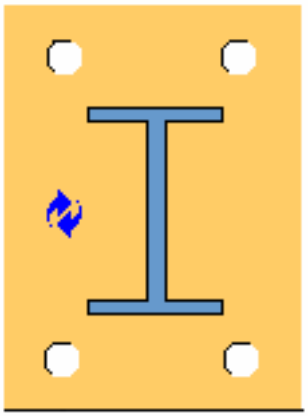
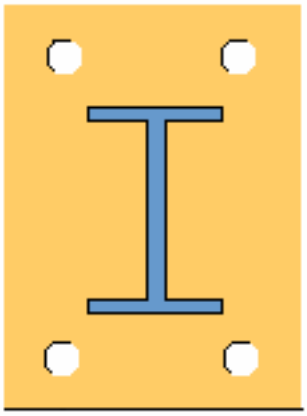
Option	Description
	Default Mounting grooves are not created. AutoDefaults can change this option.
	Mounting grooves are not created.
	Mounting grooves are created horizontally.

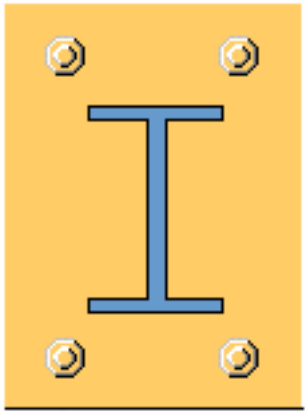
Option	Description
	Mounting grooves are created vertically for the first and last row of the bolts. For other rows of bolts, the mounting grooves are created horizontally.

Tolerance for mounting grooves

Define the tolerance for the mounting grooves in the base plates. The width of the groove is the bolt diameter + the tolerance. If you do not enter a value, the bolt tolerance value is used.

Base plate with holes or bolts

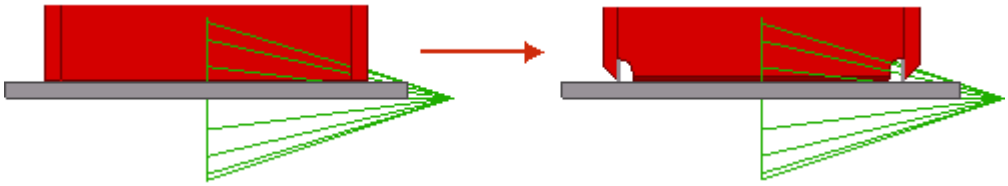
Option	Description
	Default Holes are created. AutoDefaults can change this option.
	Holes are created.

Option	Description
	Bolts are created.

Using additional components

You can use additional system or custom components to modify the column end or the base plate. For example, you can create special backing plates, weld preparations, and weld access holes for the column end.

If you use additional system or custom components, you need to manage the column end or the base plate properties in the additional component in question. When using several components, there may be multiple welds and cuts.



Option	Description
Component	Define a system or a custom component by selecting it from the component catalog.
Attributes	Enter the name of the attribute file for the selected component.
Input	Define to which parts the selected component is applied. • **Default** is same as **Base + Column**. • **Column** sets the column as the main part. Use this option for details.

Option	Description
	• **Column + Base** sets the column as the main part and the base plate as the secondary part. • **Base + Column** sets the base plate as the main part and the column as the secondary part. • **Base** sets the base plate as the main part.

Anchor rods tab

Use the **Anchor rods** tab to control the creation of different types of anchor rods.

Anchor rod dimensions

Option	Description
Rod profile	Anchor rod profile. You can add a comment about the part.
Nut profile	Nut profile.
Washer profile	Washer profile.
Plate washer	Plate washer thickness, width and height.
Cast plate	Cast plate thickness, width and height.
Grout	Grout thickness. Grouting helps you to model columns to the top of concrete parts and place the base plate correctly. It also makes it easier to dimension the detail in GA drawings. By default, no grouting is created. Select whether the grouting is created with or without slopes above or below the detail creation point. This also affects the shim plates.

Anchor rod part properties

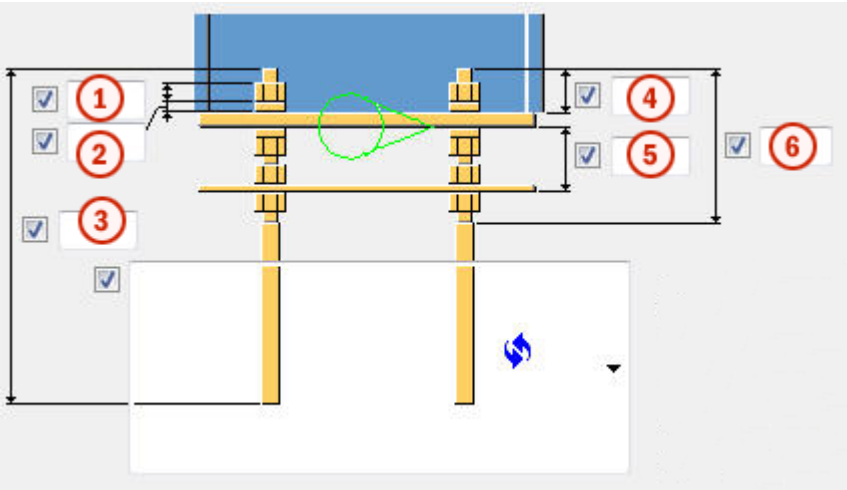
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Base plate with

Select whether to create the base plate with bolts, anchor rods, or a custom component.

By default, the base plate is created with **Bolts**.

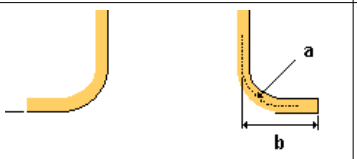
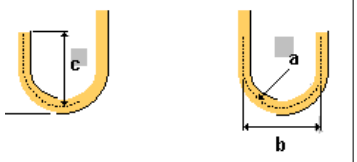
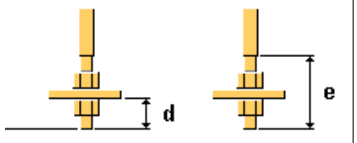
Anchor rod dimensions



	Description	Default
1	Size or the length of the nut.	`anchor rod diameter`
2	Size or the thickness of the washer.	`half of nut size`
3	Length of the anchor rod.	`500 mm`

	Description	Default
4	Length of the anchor rod above the base plate.	50 mm
5	Distance between the cast plate and the base plate.	60 mm
6	Length of the upper thread.	0 mm

Anchor rod types

Option	Description	
	Default Type 1 AutoDefaults can change this option.	
	Type 1	
	**a** Radius of the hook **b** Width of the hook	**a** = 2 x anchorbar diameter **b** = 1/5 of anchorbar length
	**a** Radius of the hook **b** Width of the hook **c** Height of the hook	**c** = same as width of the hook
	**d** Length of the anchor rod below the extra plate **e** Length of the lower thread	**d** = 2 x nut size **e** = 4 x nut size plus thickness of extra plate

Hook direction

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2
	Type 3
	Type 4

Bolting direction

NOTE You can define the bolting direction if you have created the base plate with bolts.

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1
	Bolting direction 2

Cast plate hole tolerance

Option	Description	Default
Cast plate holes tolerance	Define the tolerance of the cast plate holes.	`same as bolt tolerance`

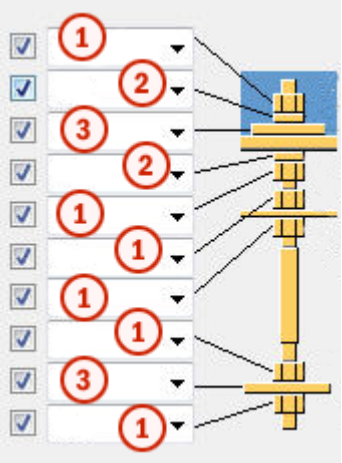
Washer hole tolerance

Option	Description
Create hole in washer	By default, a hole is not created in the washer. Tolerance of the washer hole.

Create assembly from all anchors

Define whether anchors are included in an anchor rod assembly. You can also include leveling plates into the assembly.

Create



	Description
1	Create the nut profile. You can select to create two nuts at the top of the anchor rods.
2	Create the washer profile.
3	Create the washer plate.

Anchor rod assembly

Define which parts of the anchor rod are included in the anchor rod assembly.

You can weld the washer plates above and below the base plate.

Extra plates tab

Use the **Extra plates** tab to control the placement, rotation, and type of the profiles (extra profile 1) created at the bottom of each anchor bar and the profiles (extra profile 2) that connect rows of anchor bars.

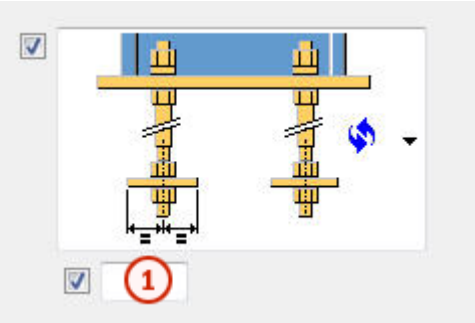
Part dimensions

Option	Description	Default
Extra profile 1	First extra profile by selecting it from the profile catalog.	`PL10 x 100`
Extra profile 2	Second extra profile by selecting it from the profile catalog.	

Part properties

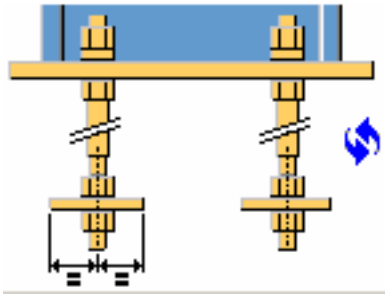
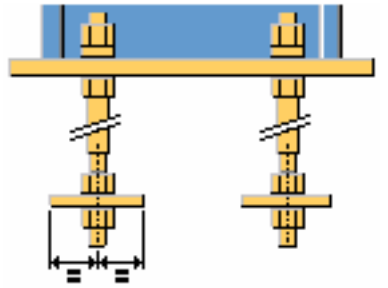
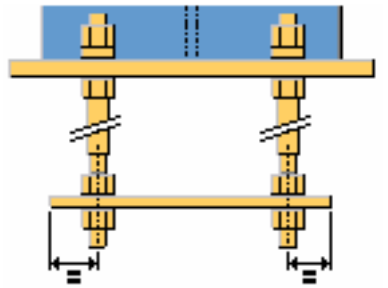
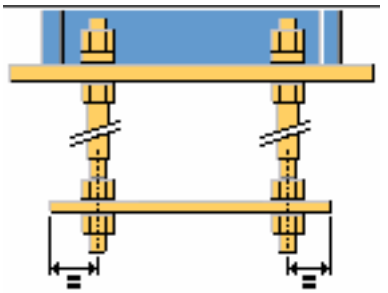
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Edge distance of extra profile 1

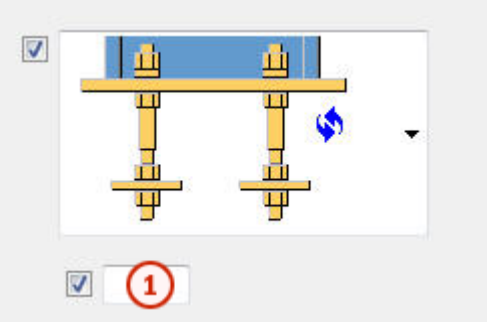


	Description	Default
1	Edge distance of extra profile 1.	50 mm

Type and direction of extra profile 1

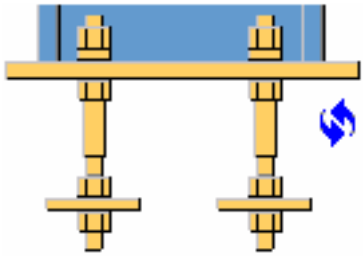
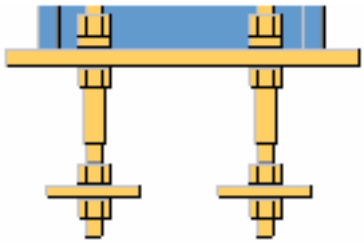
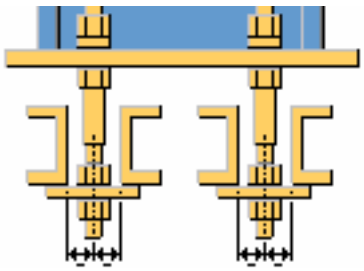
Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2
	Type 3

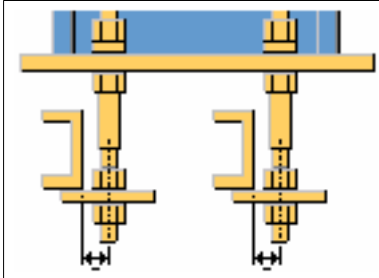
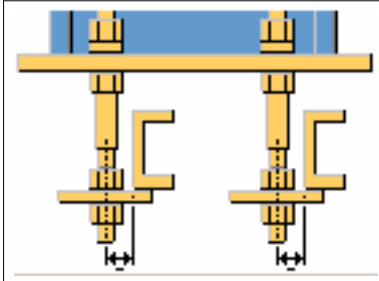
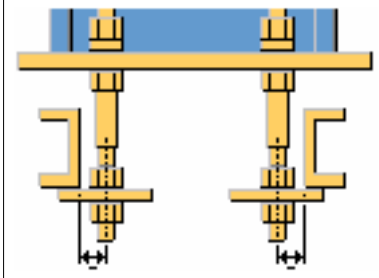
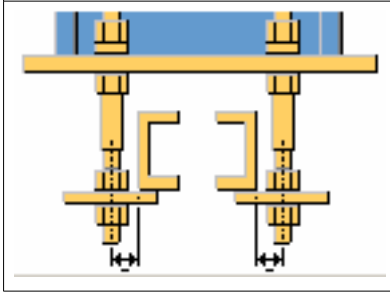
Edge distance of extra profile 2



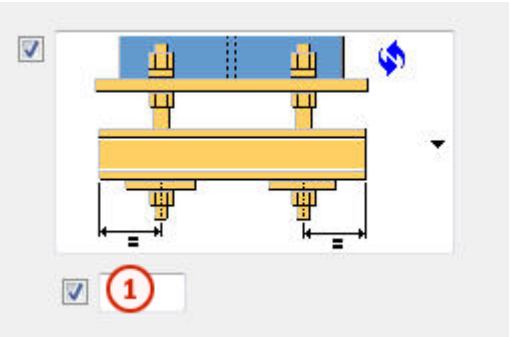
	Description	Default
1	Distance of extra profile 2 from the axis of the anchor bar.	`Half of the nut size or anchorbar diameter`

Extra profile 2 type

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2

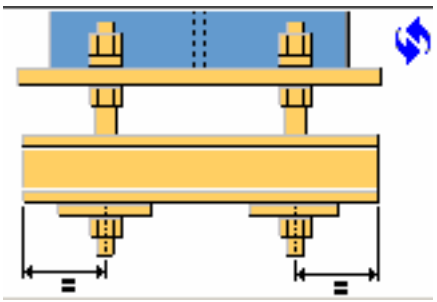
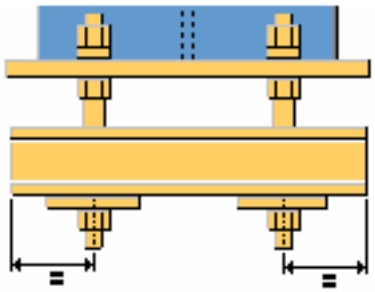
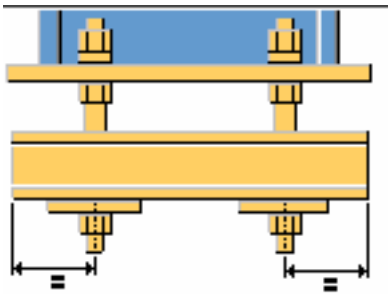
Option	Description
	Type 3
	Type 4
	Type 5
	Type 6

Length of extra profile 2



	Description	Default
1	Length of extra profile 2 from the axis of the anchor bar.	`50 mm`

Direction of extra profile 2

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2

Extra profile 1 properties

Option	Description	Default
Hole tolerance	Hole tolerance of extra profile 1.	`Same as bolt tolerance`
Circular profile height	Enter the height of a circular extra profile 1.	
Profile rotation	Profile rotation of extra profile 1.	`Front`

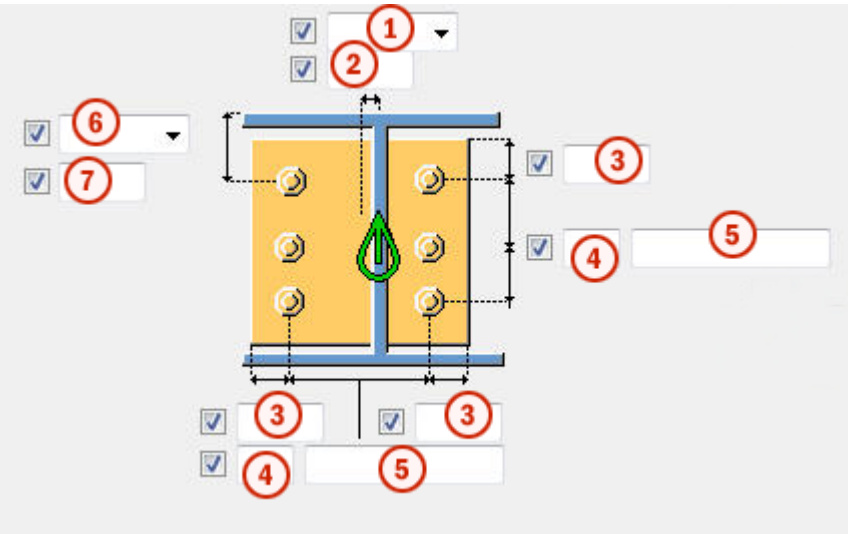
Extra profile 2 rotation

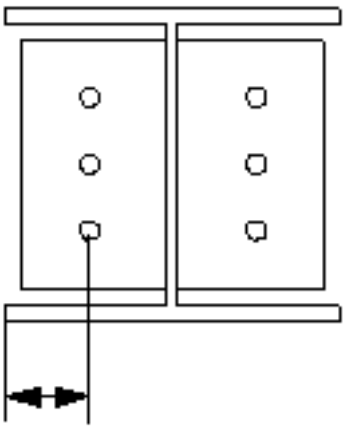
Option	Description	Default
Extra profile 2 rotation	Profile rotation of extra profile 2.	`Front`

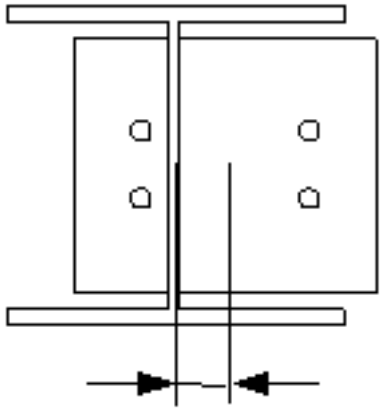
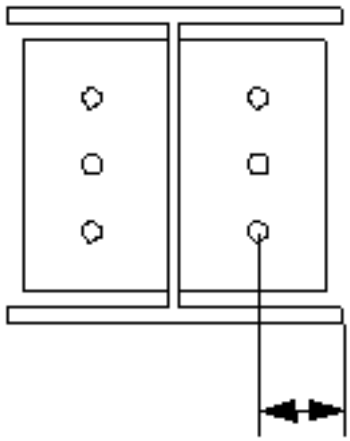
Bolts tab

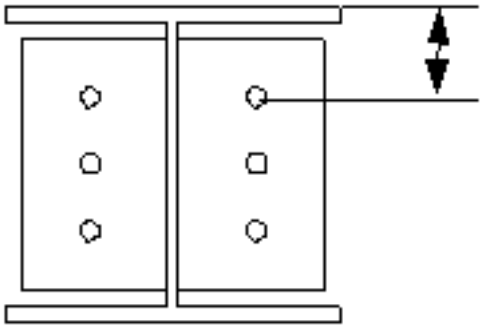
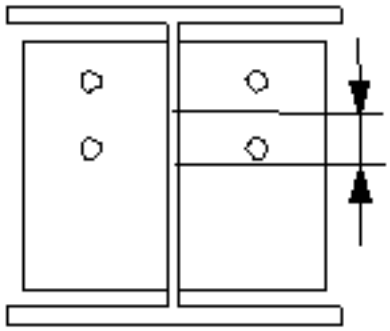
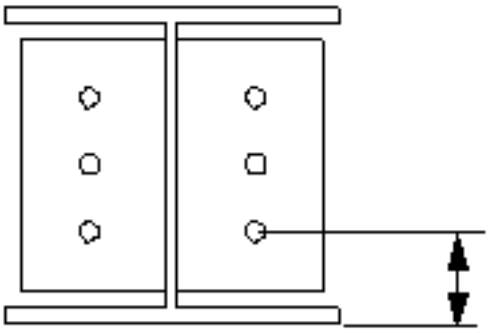
Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt. 

	Description
	• **Middle**: From the center line of the secondary part to the center line of the bolts.  • **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

	Description
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Dimension for vertical bolt group position.

Bolt basic properties

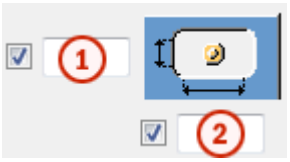
Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Slotted holes

You can define slotted, oversized, or tapped holes.



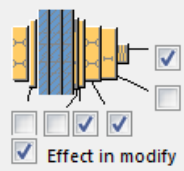
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered

Option	Description
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

U.S. Bearing plate (1044)

U.S. Bearing plate (1044) creates a bearing plate with web stiffeners for a beam, and either bolts or welded studs.

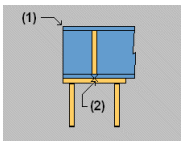
Objects created

- Bearing plate
- Stiffeners
- Bolts
- Studs
- Anchor rods
- Welds

Use for

Situation	Description
	Bearing plate is bolted to a beam. Stiffener is created.

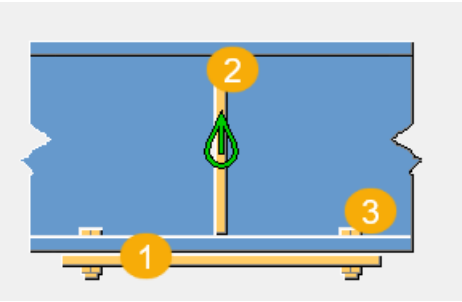
Selection order



1. Select the beam.
2. Pick a position.

   The connection is created when you pick the position.

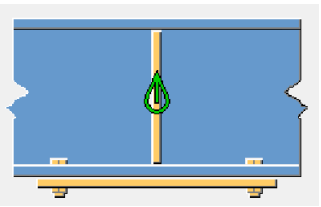
Part identification key



	Description
1	Bearing plate
2	Web stiffener
3	Bolts or welded studs

Picture tab

The **Picture** tab shows the bearing plate detail.



Plates tab

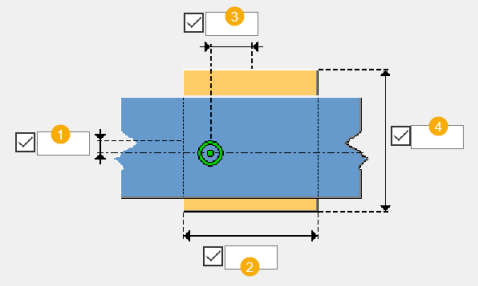
Use the **Plates** tab to define the plate dimensions and properties.

Part

Option	Description
Bearing plate	Thickness of the bearing plate.

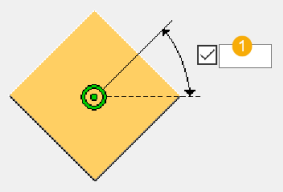
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Dimensions



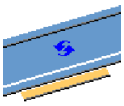
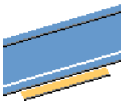
	Description
1	Vertical dimension to shift the bearing plate.
2	Length of the bearing plate. The length is calculated based on the size of the main part.
3	Horizontal dimension to shift the bearing plate.
4	Width of the bearing plate. The width is calculated based on the size of the main part.

Bearing plate rotation



	Description
1	Bearing plate rotation in degrees, measured in the plane of the plate.

Bearing plate position

Option	Description
	Default Sloped bearing plate is created. AutoDefaults can change this option.
	Sloped bearing plate is created.
	Horizontal bearing plate is created.

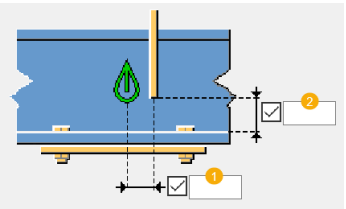
Stiffeners tab

Use the **Stiffeners** tab to define the beam web stiffener positions and properties.

Parts

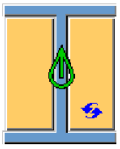
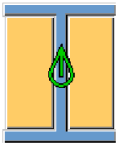
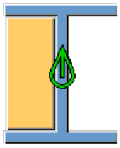
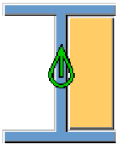
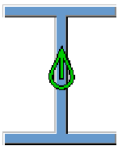
Option	Description
Stiffeners	Thickness, width and height of the stiffeners.

Stiffener offsets

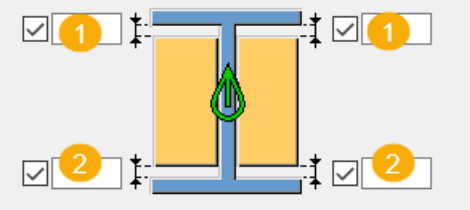


	Description
1	Stiffener horizontal offset from the point picked on the bearing plate.
2	Stiffener vertical offset from the beam web.

Stiffener location

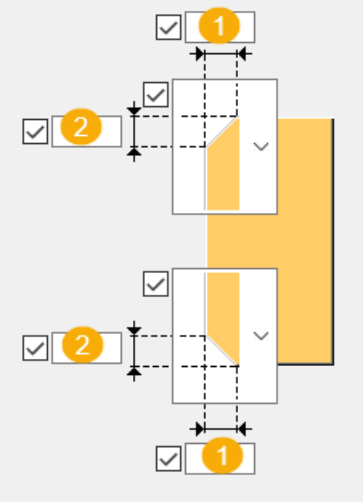
Option	Description
	Default Stiffener is created and welded to both sides of the beam. AutoDefaults can change this option.
	Stiffener is created and welded to both sides of the beam.
	Stiffener is created and welded to the far side of the beam web.
	Stiffener is created and welded to the near side of the beam web.
	Stiffeners are not created.

Vertical stiffener offset



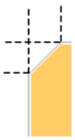
	Description
1	Stiffener offset from the main part top flange.
2	Stiffener offset from the main part bottom flange.

Stiffener chamfer dimensions



	Description
1	Horizontal dimension of the chamfer.
2	Vertical dimension of the chamfer.

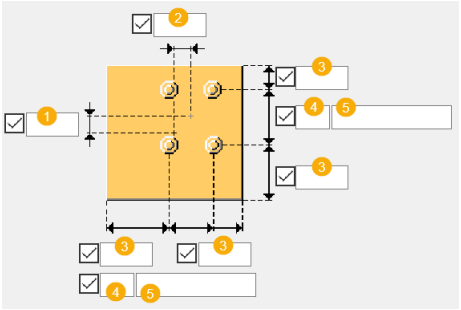
Chamfer shape

Option	Option	Description
		Default No chamfer AutoDefaults can change this option.
		No chamfer
		Line chamfer
		Convex chamfer
		Concave chamfer

Bolts tab

Use the **Bolts** tab to define the bolt group dimensions, and the bolt and stud properties.

Bolt group dimensions



	Description
1	Dimension for vertical bolt group position.
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

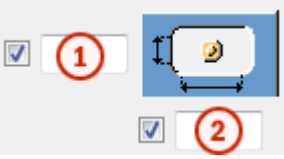
Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Stud basic properties

Option	Description
Stud size	Select the stud size.
Stud standard	Stud standard to be used inside the component.
Stud length	Select the stud length.
Site/Workshop	Location where the studs should be attached.

Slotted holes

You can define slotted, oversized, or tapped holes.



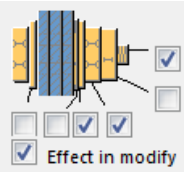
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options	

Option	Description	Default
	depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Anchor rods tab

Use the **Anchor rods** tab to define whether bolts, studs, anchor rods, or custom components are used, and their properties.

Create as

Select whether to create the bearing plate with bolts, studs, anchor rods, or a custom component.

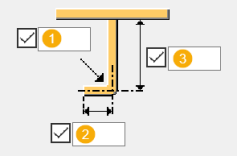
Option	Description
Bolts	Define the bolt group dimensions and bolt properties on the **Bolts** tab.
Studs	Define the stud properties on the **Bolts** tab.
Anchors	Select the anchor profile from the profile catalog.
Custom component	Select the component from the component catalog, and define the custom settings, up direction, and rotation.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Anchor bend direction

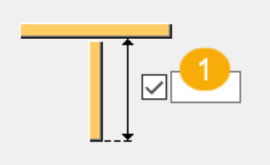
Option	Description
	Default Anchors are bent in. AutoDefaults can change this option.
	Anchors are bent in.
	Anchors are bent out.
	Anchors are not bent. Studs are created.

Bent anchor bolt length



	Description
1	Radius of the bend.
2	Bent anchor bolt length.
3	Hook length.

Stud length



	Description
1	Define the stud length.

General tab

Click the link below to find out more:
General tab

Design type tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:
Analysis tab

Welds

Click the link below to find out more:

U.S. Base plate (1047)

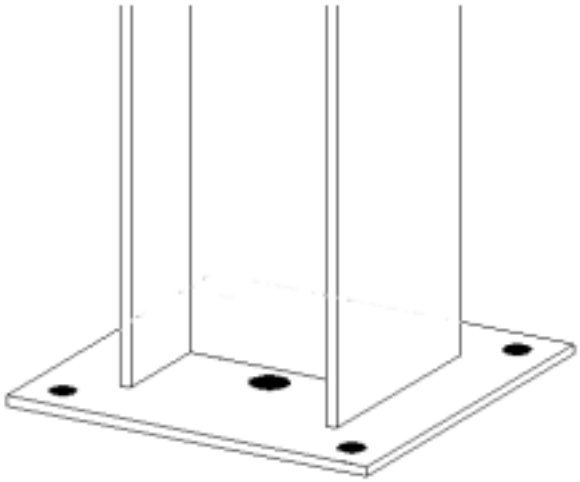
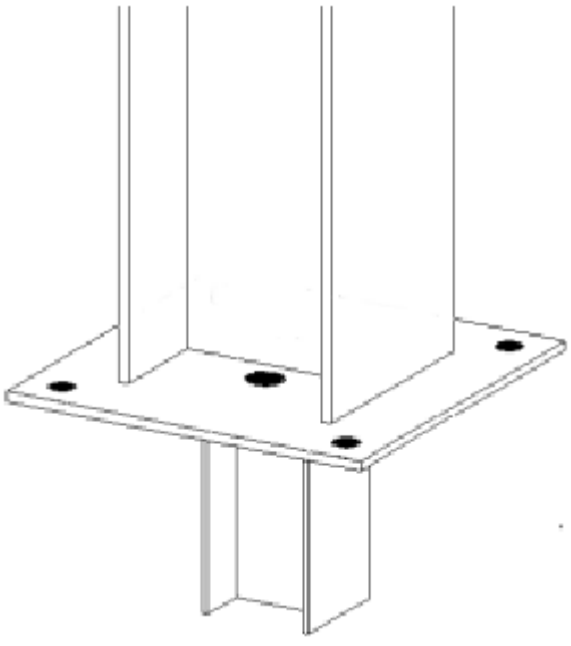
U.S. Base plate (1047) creates a base plate at the selected point on a column.

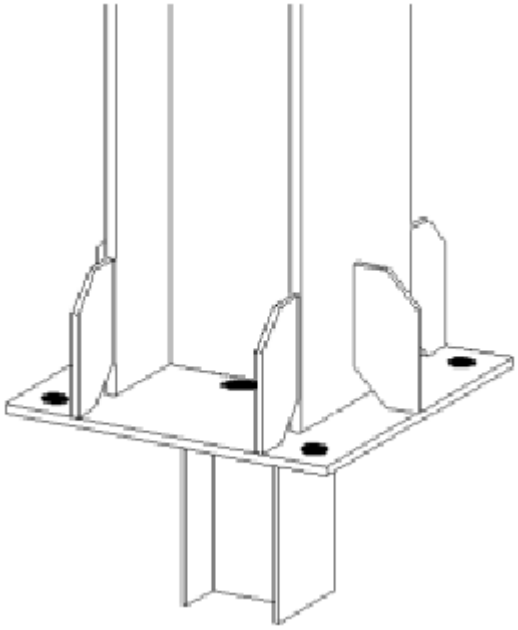
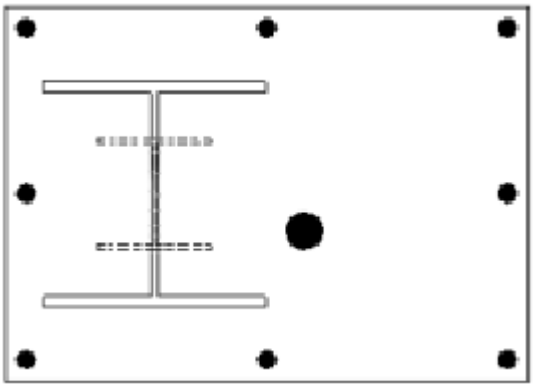
Objects created

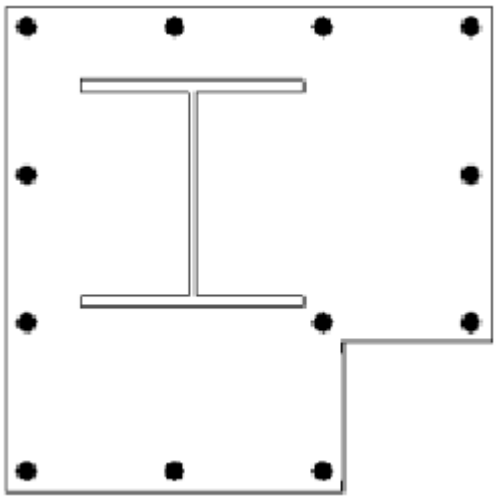
- Base plate
- Stiffeners (can only be added if the column has a W, I, or a TS profile)
- Shim plates (optional)
- Leveling plate (optional)
- Shear key (optional)
- Extra plates connecting the anchor rods

- Anchor rods
- Bolts
- Welds
- Additional component (optional)

Use for

Situation	Description
	Column base plate with a grout hole.
	Column base plate with a grout hole and a shear key.

Situation	Description
	Column base plate with a grout hole, a shear key and stiffeners.
	Offset column base plate with a shear key and a grout hole.

Situation	Description
	Column base plate with holes and chamfer.

Before you start

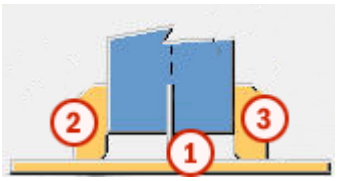
Create a column.

Selection order

1. Select the main part (column).
2. Pick a position.

   The detail is created automatically.

Part identification key

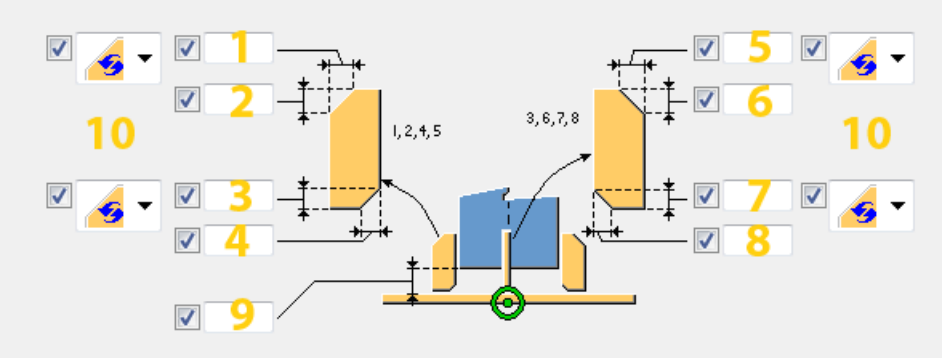


	Part
1	Base plate
2	Stiffeners 1, 2, 4, 5
3	Stiffeners 3, 6, 7, 8

Picture tab

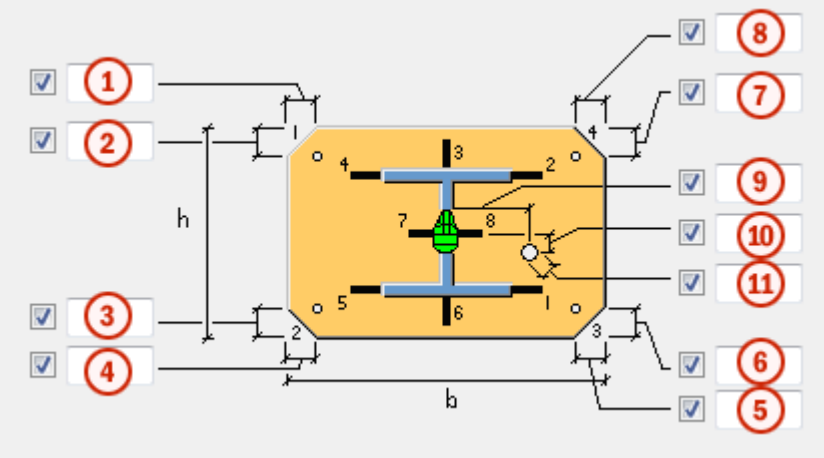
Use the **Picture** tab to control stiffener chamfering, base plate chamfers, and the location and size of the grout hole.

Stiffener chamfering



	Description
1	Top chamfer width for stiffeners 1, 2, 4, 5.
2	Top chamfer height for stiffeners 1, 2, 4, 5.
3	Bottom chamfer height for stiffeners 1, 2, 4, 5
4	Bottom chamfer width for stiffeners 1, 2, 4, 5.
5	Top chamfer width for stiffeners 3, 6, 7, 8.
6	Top chamfer height for stiffeners 3, 6, 7, 8.
7	Bottom chamfer height for stiffeners 3, 6, 7, 8.
8	Bottom chamfer width for stiffeners 3, 6, 7, 8,
9	Weld gap.
10	Select the chamfer type.

Base plate chamfering



	Description
1	Width of base plate chamfer 1.
2	Height of base plate chamfer 1.
3	Height of base plate chamfer 2.

	Description
4	Width of base plate chamfer 2.
5	Width of base plate chamfer 3.
6	Height of base plate chamfer 3.
7	Height of base plate chamfer 4.
8	Width of base plate chamfer 4.
9	Grout hole horizontal distance from the column center.
10	Grout hole vertical distance from the column center.
11	Diameter of the grout hole.

Parts tab

Use the **Parts** tab to control the dimensions of the base plate, stiffeners, leveling plate, and shim plates.

Plate

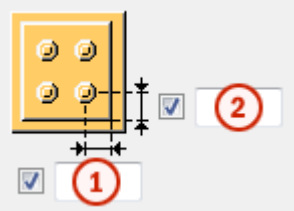
Option	Description	Default
Plate	Base plate thickness, width and height. You can add a comment about the part.	Thickness = `20 mm` Width depends on the largest cross-sectional dimension of the column. Height depends on the largest cross-sectional dimension of the column.
Stiffeners 1,2,4,5	Stiffener 1, 2, 4, 5 thickness, width and height.	Thickness depends on the column flange thickness. Width depends on the distance between the edge of the plate and the edge of the column. Height = `1.75*width`
Stiffeners 3,6,7,8	Stiffener 3, 6, 7, 8 thickness, width and height.	Thickness depends on the column flange thickness. Width depends on the distance between the

Option	Description	Default
		edge of the plate and the edge of the column. Height = `1.75*width`
Key profile	Shear key profile by selecting it from the profile catalog.	
Leveling plate	Leveling plate thickness, width, and height. You can add a comment about the part.	
Fitting plate	Shim plate thickness, width and height. Define up to three different shim plates.	
Number of fitting pl.	Number of shim plates for each thickness.	`1`
Leveling plate hole diameter	Leveling plate hole diameter.	
Key profile welded to	Define to which plate the shear key is welded.	

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

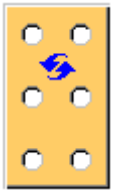
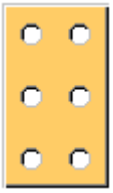
Bolt edge distances in shim plate

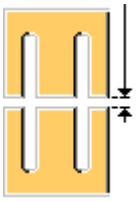
Define the bolt edge distances for shim plates. When these fields are empty, shim plates are of the same size as the base plate.



	Description	Default
1	Horizontal bolt edge distance in the shim plate.	30 mm
2	Vertical bolt edge distance in the shim plate.	30 mm

Shim plate shape

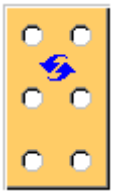
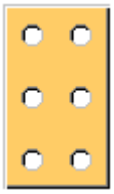
Option	Description
	Default Holes are based on the bolt group of the connection. AutoDefaults can change this option.
	Holes are based on the bolt group of the connection.
	Finger shim plate with horizontal slots. The plate can be installed from the right or the left side of the connection.
	Finger shim plate with vertical slots. The plate can be installed from the top of the connection.

Option	Description
	Two separate finger shim plates with horizontal slots.
	Two separate finger shim plates with vertical slots.

Tolerance

Define the tolerance for the slots in finger shim plates. The width of the slot is the bolt diameter + the tolerance.

Base plate mounting grooves

Option	Description
	Default Mounting grooves are not created. AutoDefaults can change this option.
	Mounting grooves are not created.
	Mounting grooves are created horizontally.
	Mounting grooves are created vertically for the first and last row of the bolts. For other rows of bolts, the mounting grooves are created horizontally.

Tolerance for mounting grooves

Define the tolerance for the mounting grooves in the base plates. The width of the groove is the bolt diameter + the tolerance. If you do not enter a value, the bolt tolerance value is used.

Parameters tab

Use the **Parameters** tab to control the plate properties and option, shear key offset, rotation and length, bolt properties and rotation of the base plate.

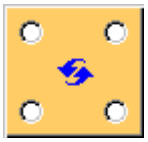
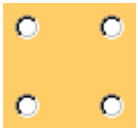
Plate properties

Option	Description
Square cut corners (1-4)	Cuts made on the corners of the base plate. If you enter a number corresponding to one of the corners of the base plate, the corner has a square cut instead of a diagonal chamfer. You can enter the numbers for one or all the corners.
Cut cast plate	Define whether base plate corner cuts are applied to the cast plate. Select **Yes** to apply the cuts.

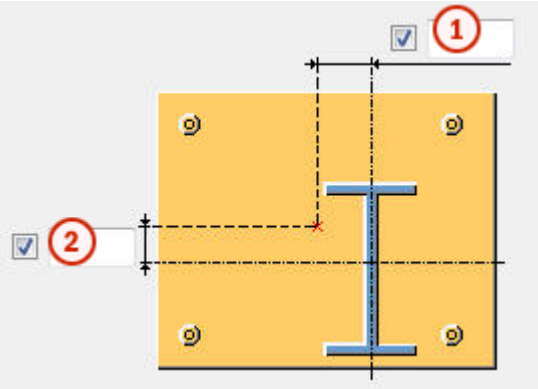
Bolt properties

Option	Description
Eliminate bolt numbers	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to down. If there are more than 10 bolts, enclose the bolt numbers in quotes (for example, to delete bolt numbers 13 and 15, enter `"13" "15"`).
Eliminate external bolts	Define whether external bolts are deleted.

Base plate with holes or bolts

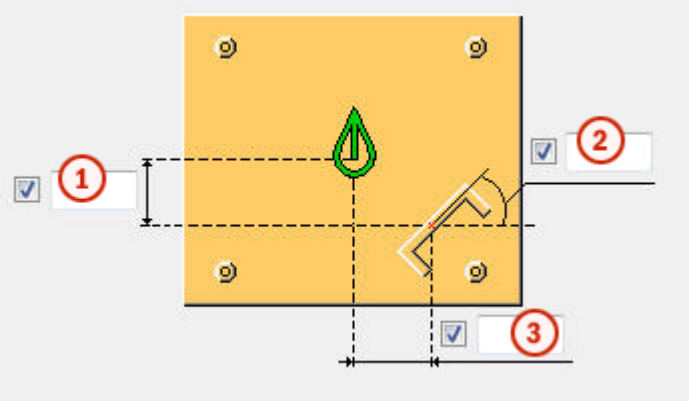
Option	Description
	Default Holes are created. AutoDefaults can change this option.
	Holes are created.
	Bolts are created.

Base plate offset



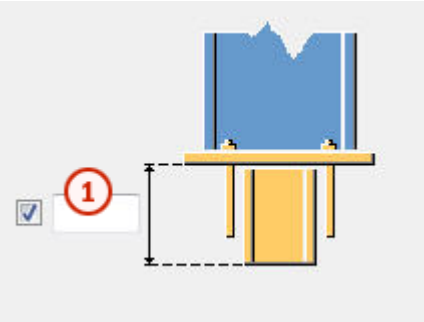
	Description
1	Base plate horizontal offset from the column center. Enter a negative value to move the base plate in the opposite direction
2	Base plate vertical offset from the column center. Enter a negative value to move the base plate in the opposite direction

Shear key offset and rotation



	Description
1	Shear key vertical offset. Enter a negative value to move the shear key in the opposite direction.
2	Shear key rotation angle (in degrees). Positive angle rotates the shear key in a counter-clockwise direction and negative angle in a clockwise direction.
3	Shear key horizontal offset. Enter a negative value to move the shear key in the opposite direction.

Shear key length

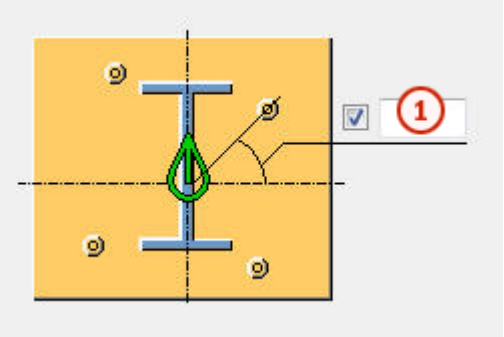


	Description
1	Shear key length to attach the shear key to the base plate. To attach a shear key to the base plate, define the shear key length and the key profile on the **Parts** tab.

Bolt group rotation

You can define the bolt group rotation around its center. To rotate the bolt group, enter the rotation angle (in degrees).

Positive angle rotates the bolts in a counter-clockwise direction and negative angle in a clockwise direction.

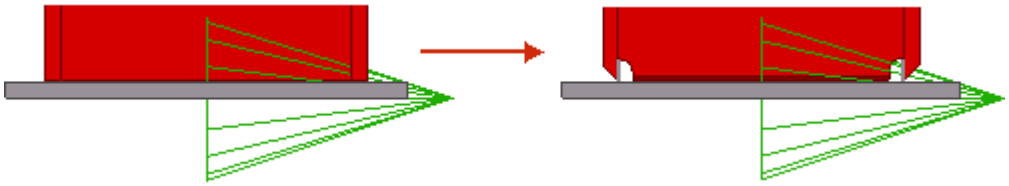


	Description
1	Bolt group rotation angle.

Using additional components

You can use additional system or custom components to modify the column end or the base plate. For example, you can create special backing plates, weld preparations, and weld access holes for the column end.

If you use additional system or custom components, you need to manage the column end or the base plate properties in the additional component in question. When using several components, there may be multiple welds and cuts.



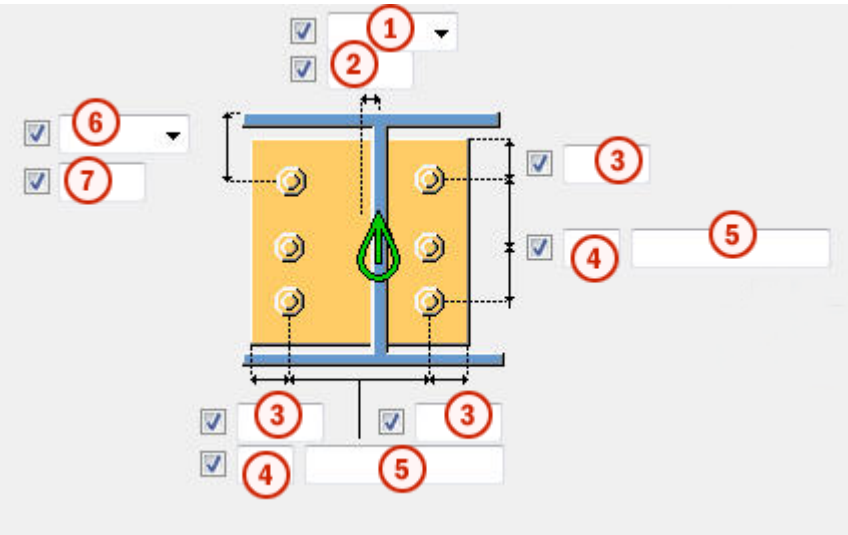
Option	Description
Component	Define a system or a custom component by selecting it from the component catalog.
Attributes	Enter the name of the attribute file for the selected component.
Input	Define to which parts the selected component is applied. • **Default** is same as **Base + Column**. • **Column** sets the column as the main part. Use this option for details. • **Column + Base** sets the column as the main part and the base plate as the secondary part.

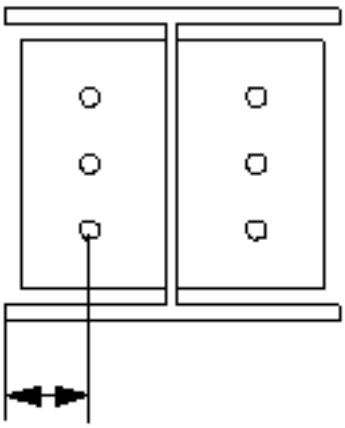
Option	Description
	• **Base + Column** sets the base plate as the main part and the column as the secondary part. • **Base** sets the base plate as the main part.

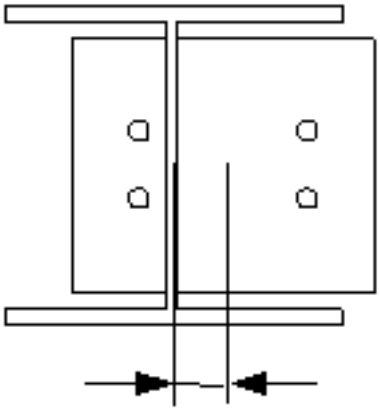
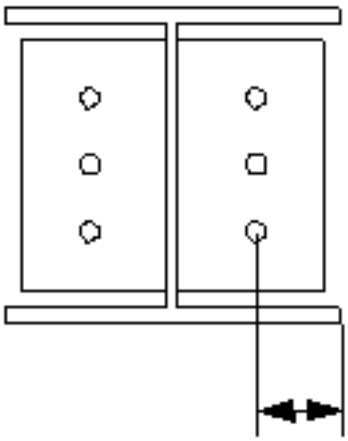
Bolts tab

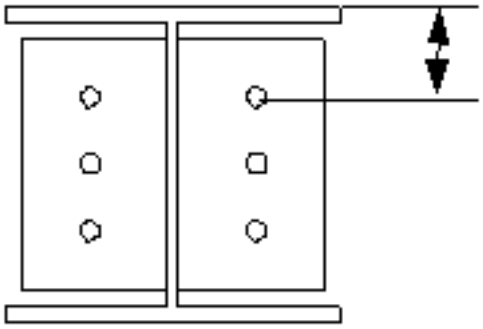
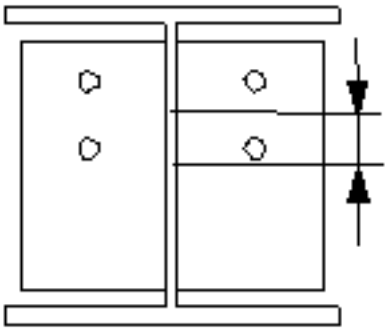
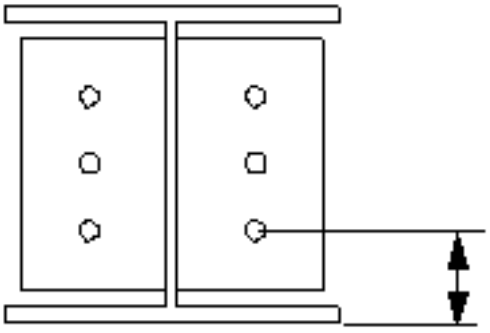
Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt. 

	Description
	• **Middle**: From the center line of the secondary part to the center line of the bolts.  • **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

	Description
6	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
7	Dimension for vertical bolt group position.

Bolt basic properties

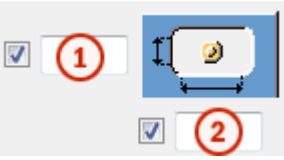
Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Slotted holes

You can define slotted, oversized, or tapped holes.



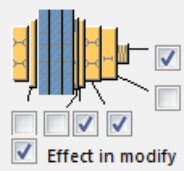
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered

Option	Description
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Stiffeners tab

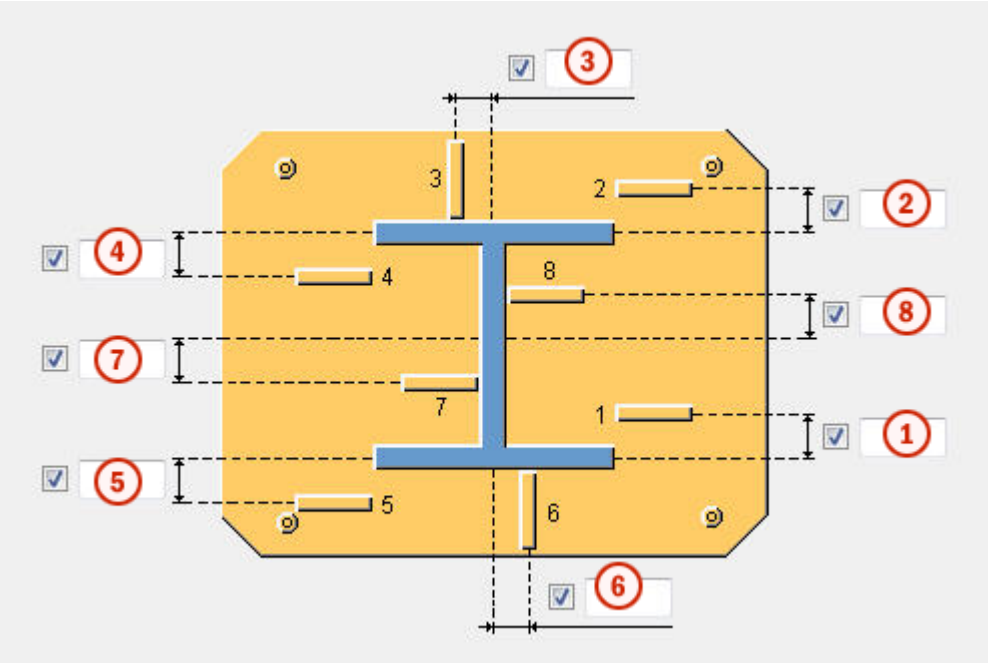
Use the **Stiffeners** tab to control stiffener positions and offsets.

Stiffener positions (1-8)

Option	Description
Stiffener Positions (1-8)	Placement of the stiffeners. Only the stiffeners whose numbers are entered in the field are attached to the column.

Stiffener offset

Stiffener offset allows the stiffeners to be moved. Enter a negative value to move the stiffener in the opposite direction.



	Description
1	Stiffener 1 offset.
2	Stiffener 2 offset.
3	Stiffener 3 offset.
4	Stiffener 4 offset.
5	Stiffener 5 offset.
6	Stiffener 6 offset.
7	Stiffener 7 offset.
8	Stiffener 8 offset.

Anchor rods tab

Use the **Anchor rods** tab to control the creation of different types of anchor rods.

Anchor rod dimensions

Option	Description
Rod profile	Anchor rod profile. You can add a comment about the part.
Nut profile	Nut profile.
Washer profile	Washer profile.
Plate washer	Plate washer thickness, width and height.
Cast plate	Cast plate thickness, width and height.
Grout	Grout thickness. Grouting helps you to model columns to the top of concrete parts and place the base plate correctly. It also makes it easier to dimension the detail in GA drawings. By default, no grouting is created. Select whether the grouting is created with or without slopes above or below the detail creation point. This also affects the shim plates.

Anchor rod part properties

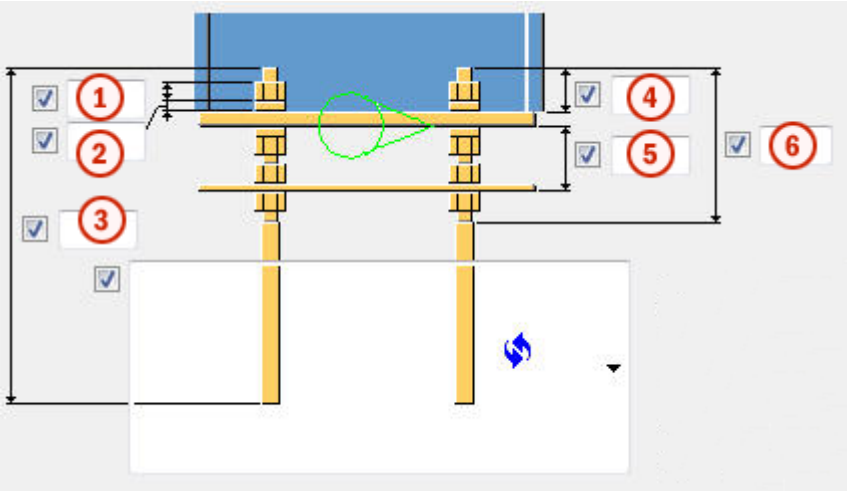
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Base plate with

Select whether to create the base plate with bolts, anchor rods, or a custom component.

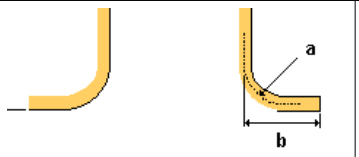
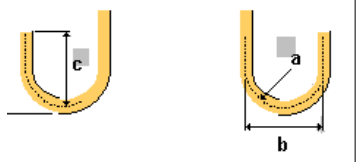
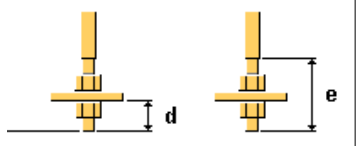
By default, the base plate is created with **Bolts**.

Anchor rod dimensions



	Description	Default
1	Size or the length of the nut.	`anchor rod diameter`
2	Size or the thickness of the washer.	`half of nut size`
3	Length of the anchor rod.	`500 mm`
4	Length of the anchor rod above the base plate.	`50 mm`
5	Distance between the cast plate and the base plate.	`60 mm`
6	Length of the upper thread.	`0 mm`

Anchor rod types

Option	Description	
	Default Type 1 AutoDefaults can change this option.	
	Type 1	
	**a** Radius of the hook **b** Width of the hook	**a** = `2*anchor bar diameter` **b** = `1/5 of anchor bar length`
	**a** Radius of the hook **b** Width of the hook **c** Height of the hook	**c** = `same as width of the hook`
	**d** Length of the anchor rod below the extra plate **e** Length of the lower thread	**d** = `2*nut size` **e** = `4*nut size plus thickness of extra plate`

Hook direction

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2
	Type 3
	Type 4

Bolting direction

NOTE You can define the bolting direction if you have created the base plate with bolts.

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1
	Bolting direction 2

Cast plate hole tolerance

Option	Description	Default
Cast plate holes tolerance	Tolerance of the cast plate holes.	`same as bolt tolerance`

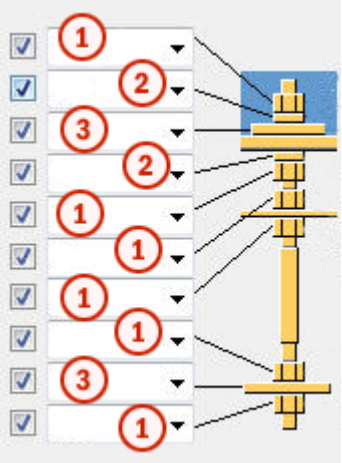
Washer hole tolerance

Field	Description
Create hole in washer	By default, a hole is not created in the washer. Tolerance of the washer hole.

Create assembly from all anchors

Define whether anchors are included in an anchor rod assembly. You can also include leveling plates into the assembly.

Create



	Description
1	Create the nut profile. You can select to create two nuts at the top of the anchor rods.
2	Create the washer profile.
3	Create the washer plate.

Anchor rod assembly

Define which parts of the anchor rod are included in the anchor rod assembly.

You can weld the washer plates above and below the base plate.

Extra plates tab

Use the **Extra plates** tab to control the placement, rotation, and type of the profiles (extra profile 1) created at the bottom of each anchor bar and the profiles (extra profile 2) that connect rows of anchor bars.

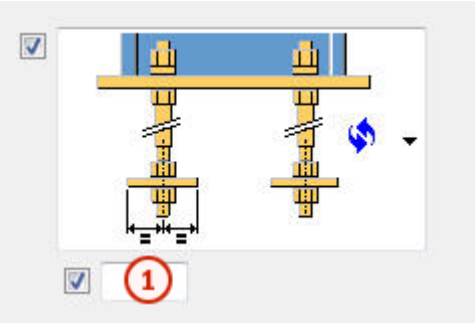
Part dimensions

Option	Description	Default
Extra profile 1	Define the first extra profile by selecting it from the profile catalog.	`PL10*100`
Extra profile 2	Define the second extra profile by selecting it from the profile catalog.	

Part properties

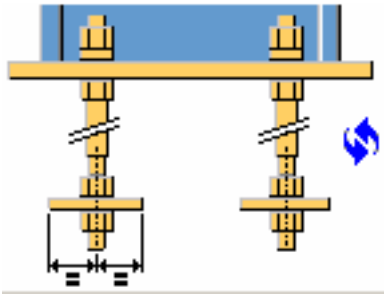
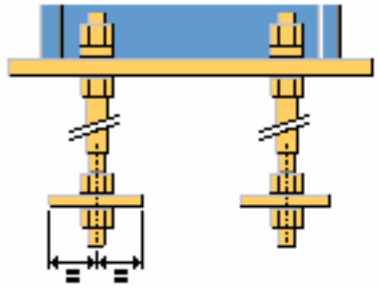
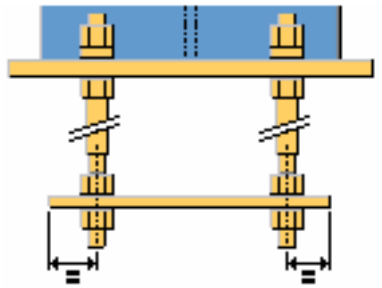
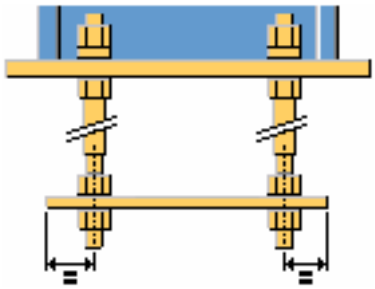
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Edge distance of extra profile 1

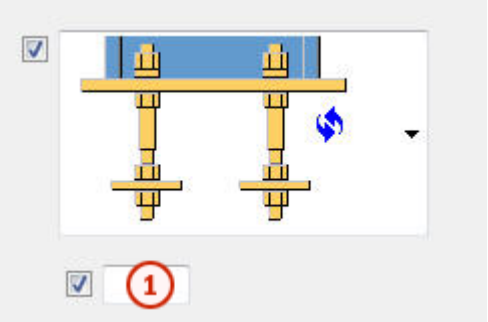


	Description	Default
1	Edge distance of extra profile 1.	50 mm

Type and direction of extra profile 1

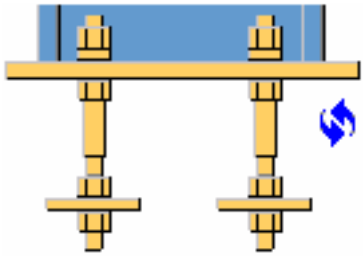
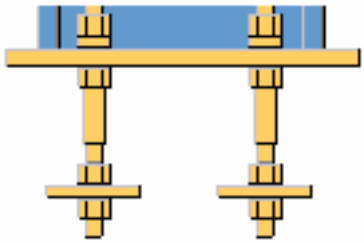
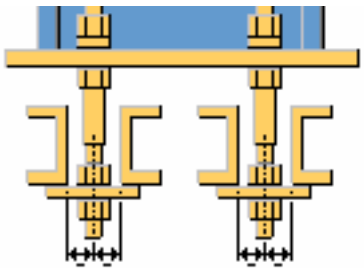
Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2
	Type 3

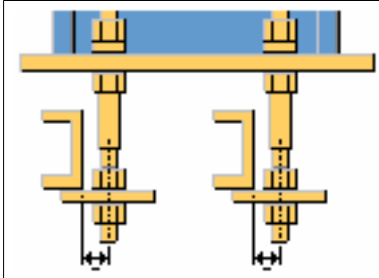
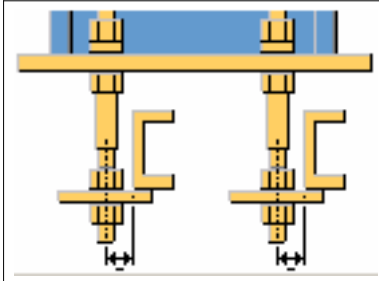
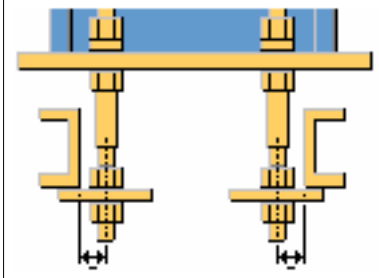
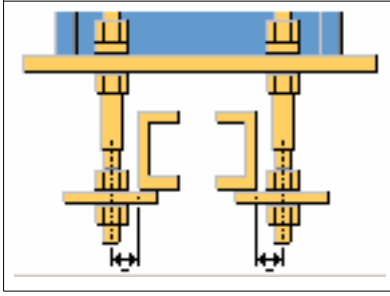
Edge distance of extra profile 2



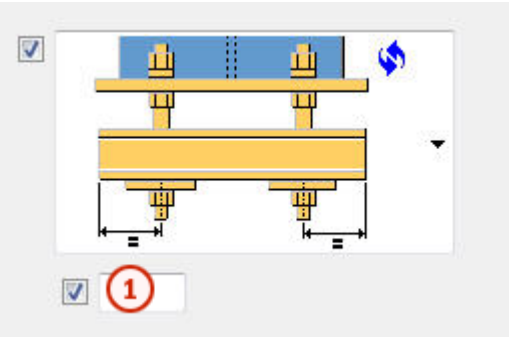
	Description	Default
1	Distance of extra profile 2 from the axis of the anchor bar.	`Half of the nut size or anchor bar diameter`

Extra profile 2 type

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2

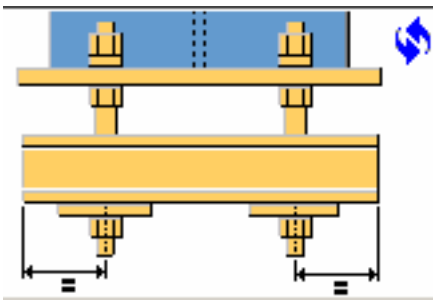
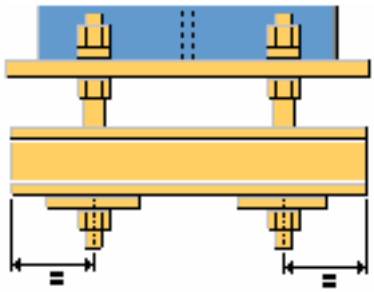
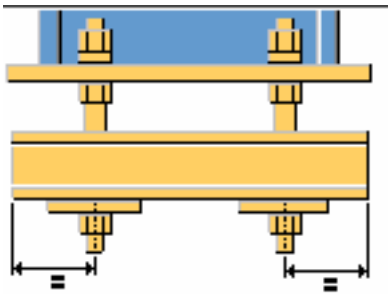
Option	Description
	Type 3
	Type 4
	Type 5
	Type 6

Length of extra profile 2



	Description	Default
1	Length of extra profile 2 from the axis of the anchor bar.	`50 mm`

Direction of extra profile2

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2

Extra profile 1 properties

Option	Description	Default
Hole tolerance	Hole tolerance of extra profile 1.	`Same as bolt tolerance`
Circular profile height	Enter the height of a circular extra profile 1.	
Profile rotation	Profile rotation of extra profile 1.	`Front`

Extra profile 2 rotation

Option	Description	Default
Extra profile 2 rotation	Profile rotation of extra profile 2.	`Front`

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Create welds

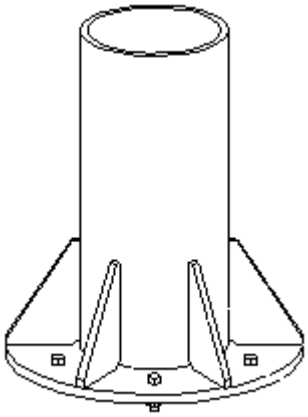
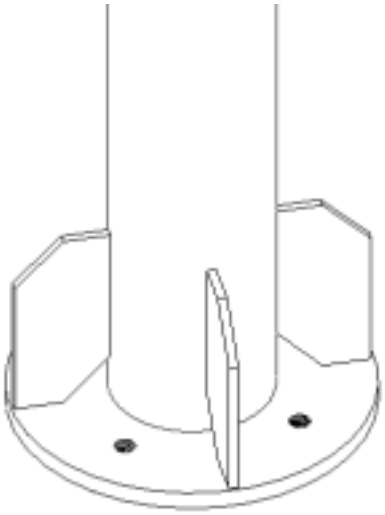
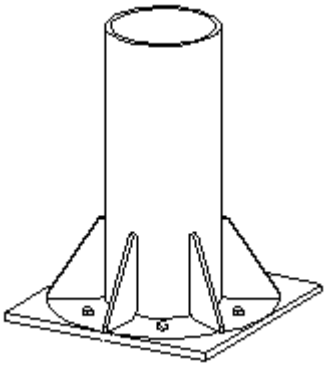
Circular base plates (1052)

Circular base plates (1052) creates a base plate that is connected to a circular column.

Objects created

- Circular base plate
- Tube stiffeners
- Extra plates connecting the anchor rods
- Anchor rods
- Bolts
- Welds
- Additional component (optional)

Use for

Situation	Description
	Circular base plate
	Circular base plate
	Square base plate

Before you start

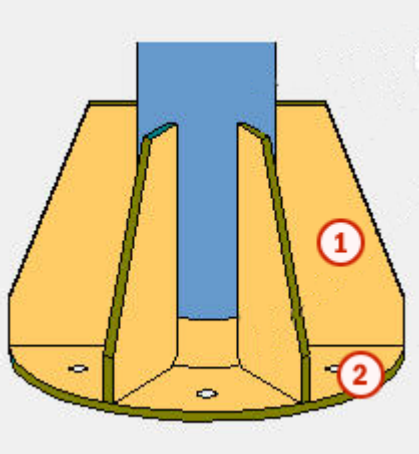
Create a circular column.

Selection order

1. Select the main part (column).
2. Pick a position.

   The detail is created automatically.

Part identification key

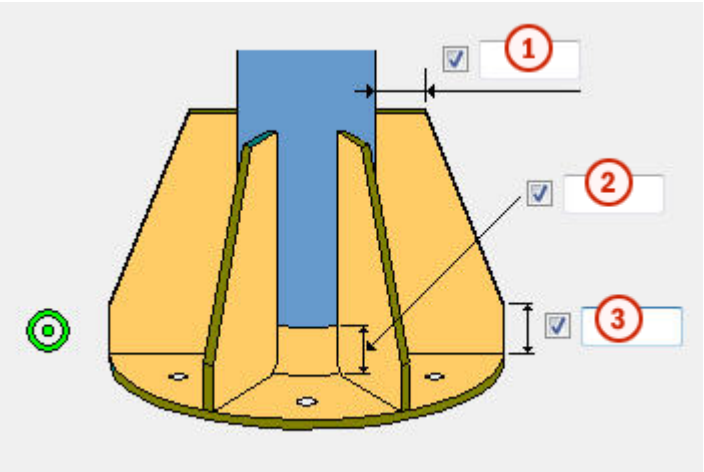


	Part
1	Stiffener
2	Base plate

Picture tab

Use the **Picture** tab to control the geometry of **Circular base plates (1052)**.

Dimensions



	Description	Default
1	Width of the top part of the tube stiffener.	20 mm
2	Weld gap.	0 mm
3	Height of the end part of the tube stiffener.	30 mm

Parts tab

Use the **Parts** tab to control the dimensions of the circular base plate and the tube stiffeners.

Plate

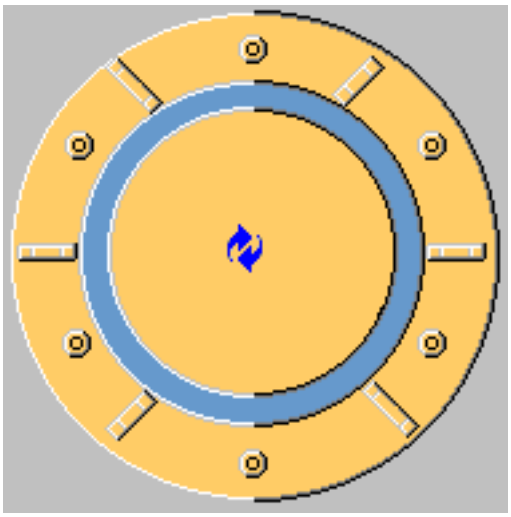
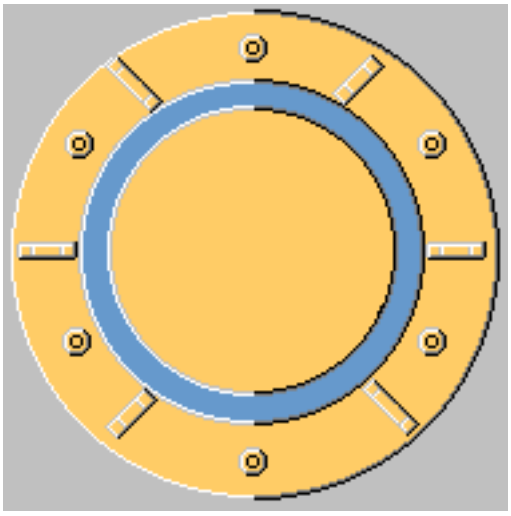
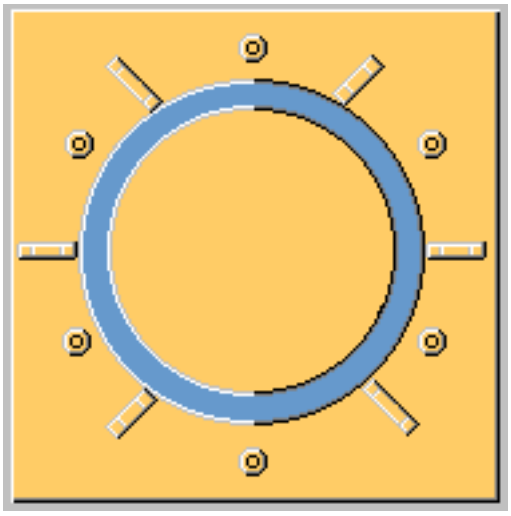
Option	Description	Default
Circular base plate	Circular base plate thickness and width.	Thickness = 30 mm Width depends on the tube diameter.
Tube stiffeners	Tube stiffener thickness, width and height.	Thickness = 10 mm Height = 200 mm

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

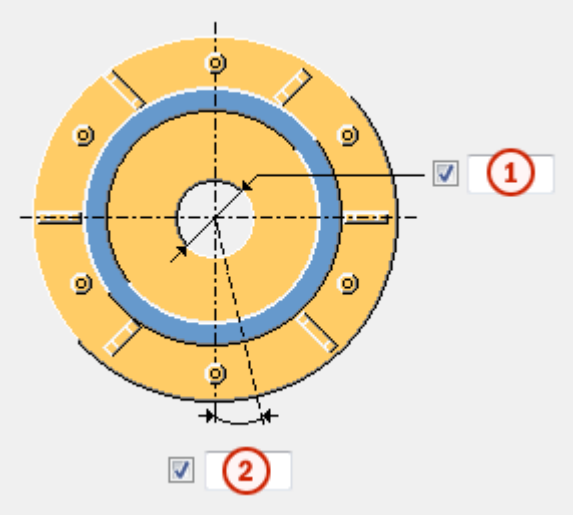
Parameters tab

Use the **Parameters** tab to control the shape of the base plate, diameter of the inner plate, bolt angle, the overlap, and chamfer dimension.

Base plate shape

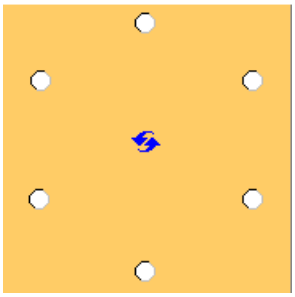
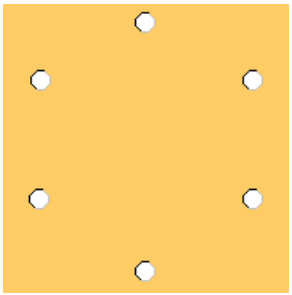
Option	Description
	Default Round base plate AutoDefaults can change this option.
	Round base plate
	Square base plate

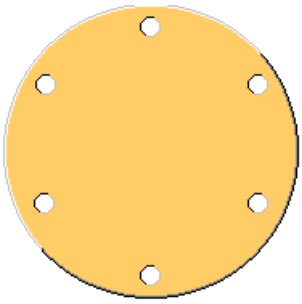
Inner plate diameter and bolt angle



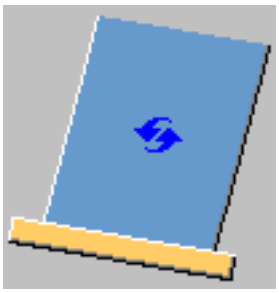
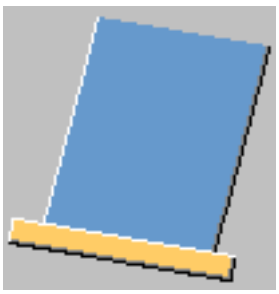
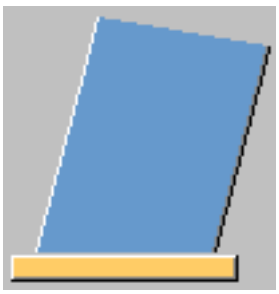
	Description
1	Base plate inner diameter.
2	Bolt angle (in degrees).

Cast plate shape

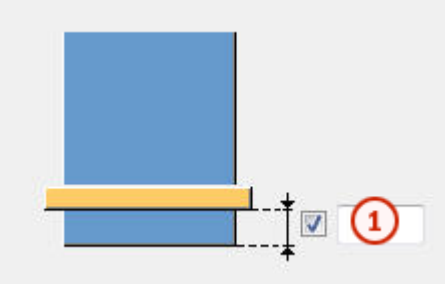
Option	Description
	Default Square cast plate AutoDefaults can change this option.
	Square cast plate

Option	Description
	Round cast plate

Base plate position

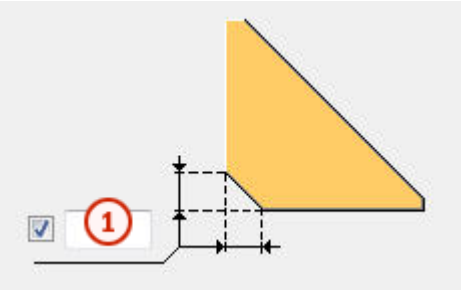
Option	Description
	Default Base plate is perpendicular to the column. AutoDefaults can change this option.
	Base plate is perpendicular to the column.
	Base plate is not perpendicular to the column.

Overlap



	Description	Default
1	Distance between the column flange and the plate flange.	0 mm

Chamfer dimension

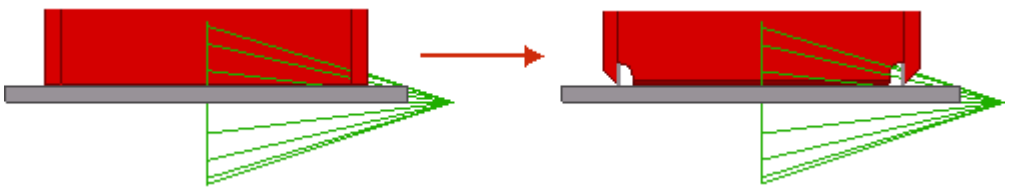


	Description	Default
1	Chamfer dimension.	10 mm

Using additional components

You can use additional system or custom components to modify the column end or the base plate. For example, you can create special backing plates, weld preparations, and weld access holes for the column end.

If you use additional system or custom components, you need to manage the column end or the base plate properties in the additional component in question. When using several components, there may be multiple welds and cuts.

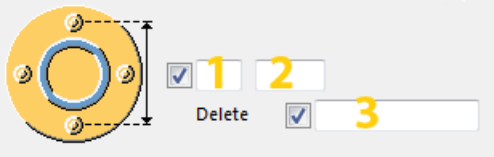


Option	Description
Component	Define a system or a custom component by selecting it from the component catalog.
Attributes	Enter the name of the attribute file for the selected component.
Input	Define to which parts the selected component is applied. • **Default** is same as **Base + Column**. • **Column** sets the column as the main part. Use this option for details. • **Column + Base** sets the column as the main part and the base plate as the secondary part. • **Base + Column** sets the base plate as the main part and the column as the secondary part. • **Base** sets the base plate as the main part.

Bolts tab

Use the **Bolts** tab to control the bolt properties.

Bolt positions



	Description
1	Number of bolts.
2	Spacing between bolts.
3	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to down.

Bolt basic properties

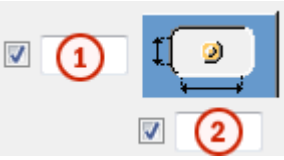
Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Slotted holes

You can define slotted, oversized, or tapped holes.



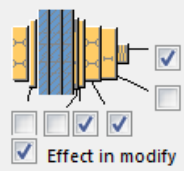
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Anchor rods tab

Use the **Anchor rods** tab to control the creation of different types of anchor rods.

Anchor rod dimensions

Option	Description
Rod profile	Anchor rod profile. You can add a comment about the part.
Nut profile	Nut profile.
Washer profile	Washer profile.
Plate washer	Plate washer thickness, width and height.
Cast plate	Cast plate thickness, width and height.
Grout	Grout thickness. Grouting helps you to model columns to the top of concrete parts and place the base plate correctly. It also makes it easier to dimension the detail in GA drawings. By default, no grouting is created. Select whether the grouting is created with or without slopes above or below the detail creation point. This also affects the shim plates.

Anchor rod part properties

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

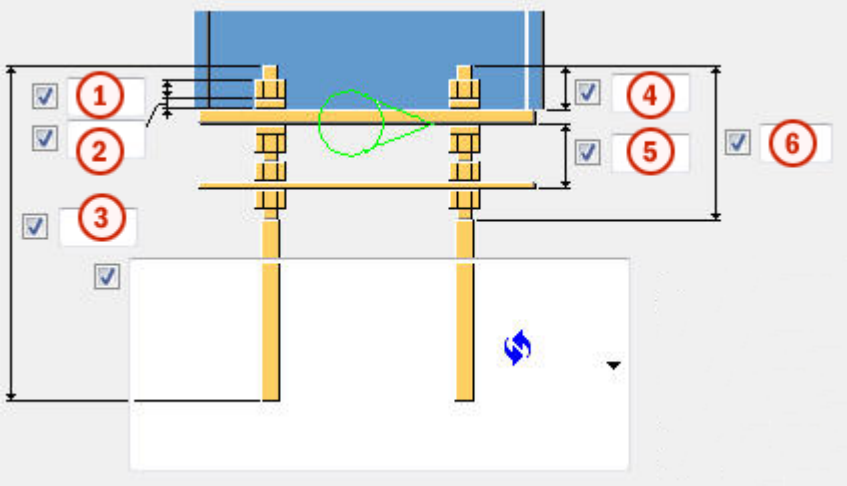
Option	Description	Default
Finish	Describes how the part surface has been treated.	

Base plate with

Select whether to create the base plate with bolts, anchor rods, or a custom component.

By default, the base plate is created with **Bolts**.

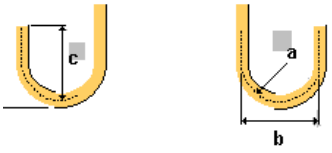
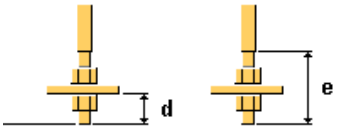
Anchor rod dimensions



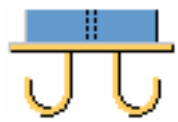
	Description	Default
1	Size or the length of the nut.	`anchor rod diameter`
2	Size or the thickness of the washer.	`half of nut size`
3	Length of the anchor rod.	`500 mm`
4	Length of the anchor rod above the base plate.	`50 mm`
5	Distance between the cast plate and the base plate.	`60 mm`
6	Length of the upper thread.	`0 mm`

Anchor rod types

Option	Description	
	Default Type 1 AutoDefaults can change this option.	

Option	Description	
	Type 1	
	a Radius of the hook **b** Width of the hook	**a** = `2*anchor bar diameter` **b** = `1/5 of anchor bar length`
	**a** Radius of the hook **b** Width of the hook **c** Height of the hook	**c** = `same as width of the hook`
	**d** Length of the anchor rod below the extra plate **e** Length of the lower thread	**d** = `2*nut size` **e** = `4*nut size plus thickness of extra plate`

Hook direction

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2
	Type 3

Option	Description
	Type 4

Bolting direction

NOTE You can define the bolting direction when you create the base plate with bolts.

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1
	Bolting direction 2

Cast plate hole tolerance

Option	Description	Default
Cast plate holes tolerance	Tolerance of the cast plate holes.	`same as bolt tolerance`

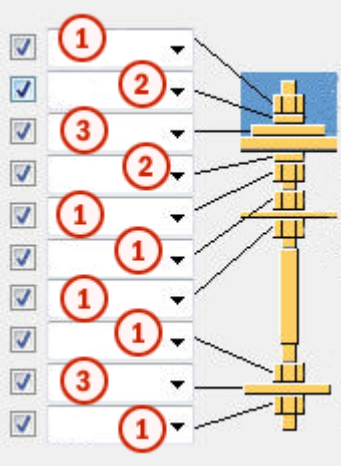
Washer hole tolerance

Option	Description
Create hole in washer	By default, a hole is not created in the washer. Tolerance of the washer hole.

Plate washer placing

Option	Description
Plate washers placed along circle	Select whether the plate washers are placed along the circle. Placing the washers along the circle distributes the loads evenly.

Create



	Description
1	Create the nut profile.
2	Create the washer profile.
3	Create the washer plate.

Anchor rod assembly

Define which parts of the anchor rod are included in the anchor rod assembly.

You can weld the washer plates above and below the base plate.

Extra plates tab

Use the **Extra plates** tab to control the placement, rotation, and type of the profiles (extra profile 1) created at the bottom of each anchor bar and the profiles (extra profile 2) that connect rows of anchor bars.

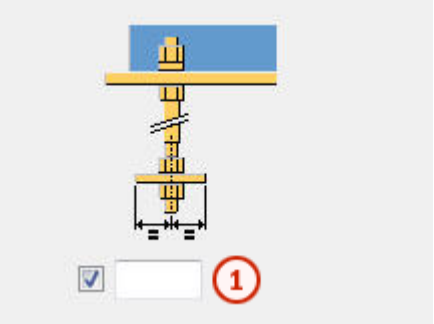
Part dimensions

Option	Description	Default
Extra profile 1	Select the first extra profile from the profile catalog.	`PL10*100`
Extra profile 2	Select the second extra profile from the profile catalog.	

Part properties

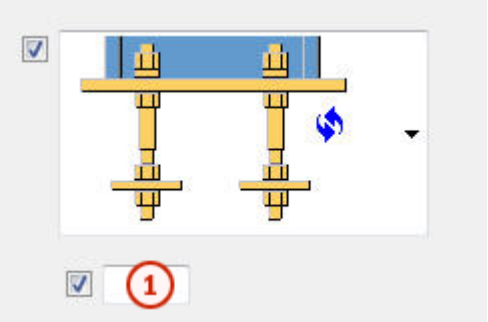
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

Edge distance of extra profile 1



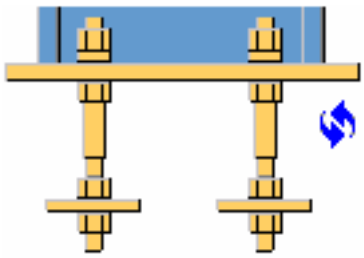
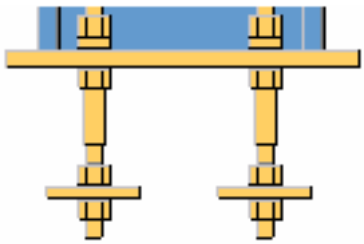
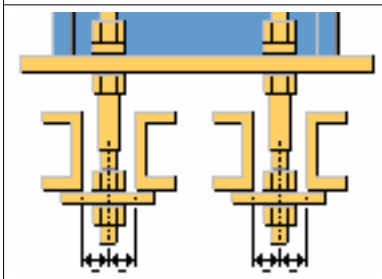
	Description	Default
1	Edge distance of extra profile 1.	50 mm

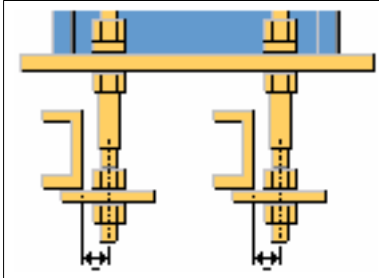
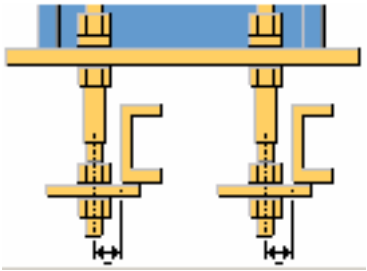
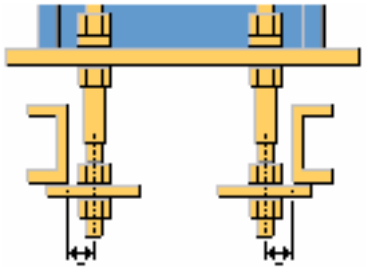
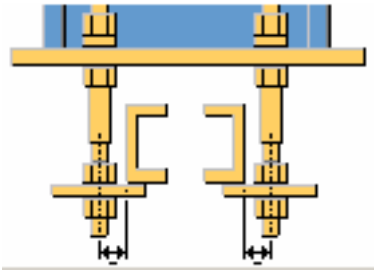
Edge distance of extra profile 2



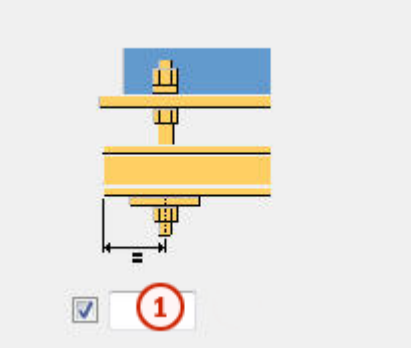
	Description	Default
1	Distance of extra profile 2 from the axis of the anchor bar.	`Half of nut size or anchor bar diameter`

Extra profile 2 type

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2

Option	Description
	Type 3
	Type 4
	Type 5
	Type 6

Length of extra profile 2



	Description	Default
1	Length of extra profile 2 from the axis of the anchor bar.	`50 mm`

Extra profile 1 properties

Option	Description	Default
Hole tolerance	Hole tolerance of extra profile 1.	`Same as bolt tolerance`
Circular profile height	Enter the height of a circular extra profile 1.	
Profile rotation	Profile rotation of extra profile 1.	`Front`

Extra profile 2 rotation

Option	Description	Default
Extra profile 2 rotation	Profile rotation of extra profile 2.	`Front`

General tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Welds

Click the link below to find out more:

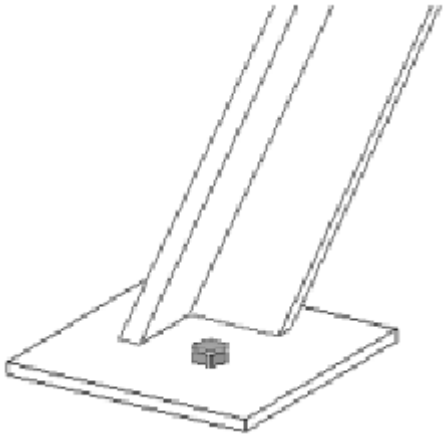
Base plate (1053)

Base plate (1053) creates a base plate for tower legs. The base plate is horizontal even if the column is sloped.

Objects created

- Base plate
- Bolts
- Welds
- Additional component (optional)

Use for

Situation	Description
	Horizontal base plate connected to sloped tower leg

Before you start

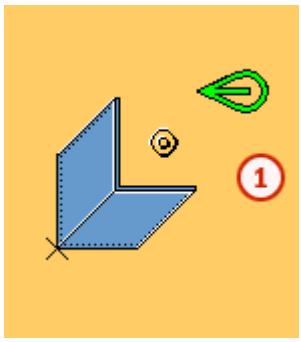
Create a column or a beam.

Selection order

1. Select the main part (column or beam).
2. Pick a position.

   The detail is created automatically.

Part identification key

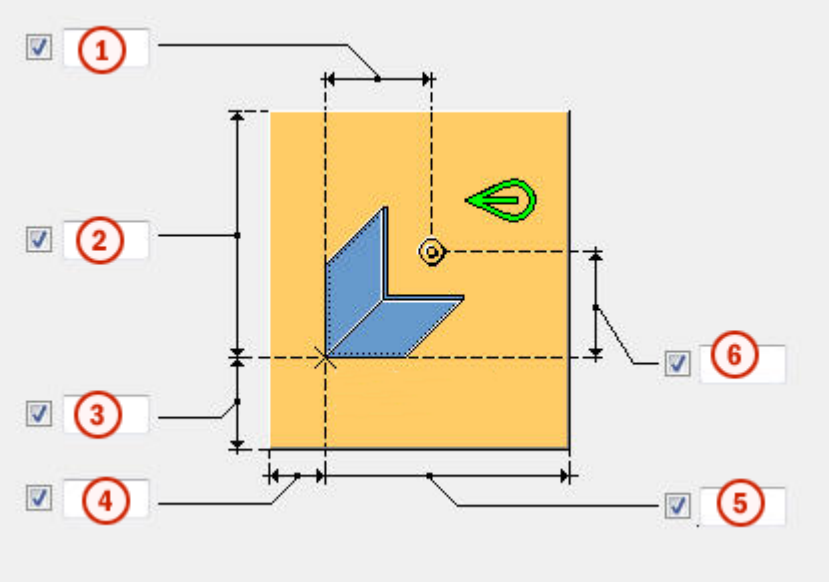


Number	Part
1	Base plate

Picture tab

Use the **Picture** tab to control the position of the base plate.

Dimensions



	Description	Default
1	Horizontal distance between the reference point and the bolt.	150 mm
2	Vertical distance between the reference point and upper edge of the base plate.	300 mm
3	Vertical distance between the reference point and the lower edge of the base plate.	200 mm
4	Horizontal distance between the reference point and the left edge of the base plate.	200 mm
5	Horizontal distance between the reference point and the right edge of the base plate.	300 mm
6	Vertical distance between the reference point and the bolt.	150 mm

Parts tab

Use the **Parts** tab to control the dimensions of the base plate.

Plate

Option	Description	Default
Plate	Thickness, width, and height of the base plate.	Thickness = 10 mm Width = 250 mm Height =250 mm

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

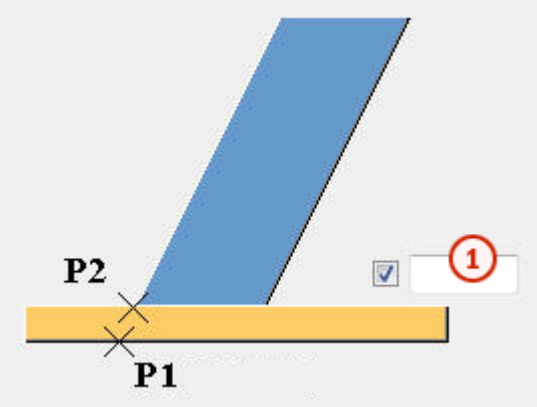
Use the **Parameters** tab to control the position of the reference point and the bolt locations.

Reference point

Define the plate dimensions and bolt locations using a reference point. The point is located at the column end angle.

The picked point locates either on the top (**P2**) or the bottom of the base plate (**P1**).

Weld gap

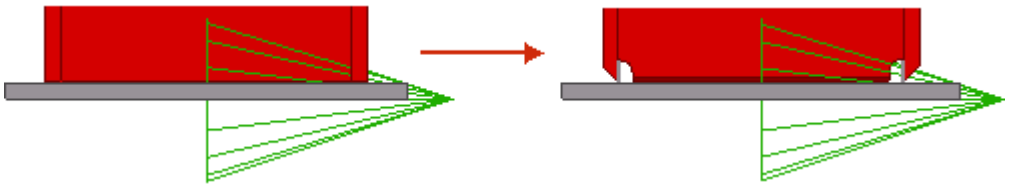


	Description
1	Gap between the base plate and the column.

Using additional components

You can use additional system or custom components to modify the column end or the base plate. For example, you can create special backing plates, weld preparations, and weld access holes for the column end.

If you use additional system or custom components, you need to manage the column end or the base plate properties in the additional component in question. When using several components, there may be multiple welds and cuts.

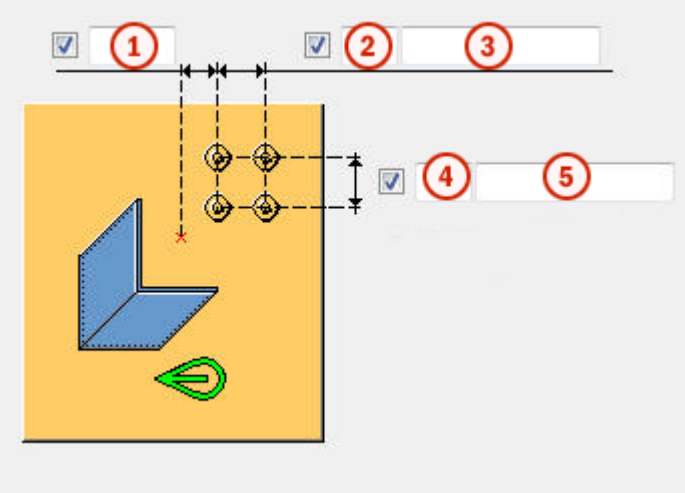


Option	Description
Component	Define a system or a custom component by selecting it from the component catalog.
Attributes	Enter the name of the attribute file for the selected component.
Input	Define to which parts the selected component is applied. • **Default** is same as **Base + Column**. • **Column** sets the column as the main part. Use this option for details. • **Column + Base** sets the column as the main part and the base plate as the secondary part. • **Base + Column** sets the base plate as the main part and the column as the secondary part. • **Base** sets the base plate as the main part.

Bolts tab

Use the **Bolts** tab to control the bolt properties.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.
2	Number of bolts in the horizontal dimension.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Number of bolts in the vertical dimension.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

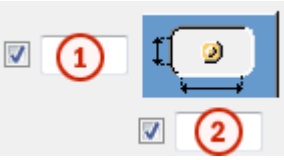
Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Slotted holes

You can define slotted, oversized, or tapped holes.

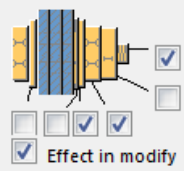


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Create as

Use this option to switch between bolts and custom components.

Select the custom component from the **Applications & components** catalog and define the custom settings, up direction, rotation, and anchor length.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

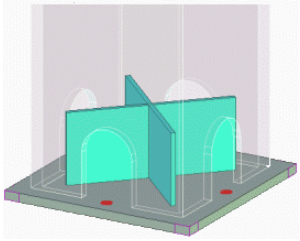
Box column base plate (1066)

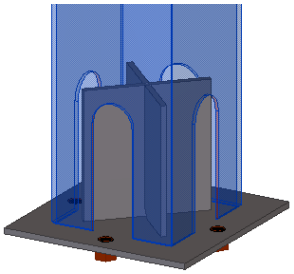
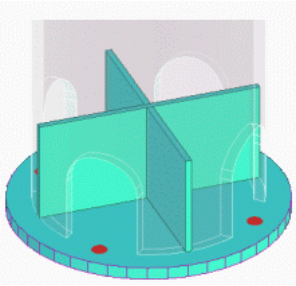
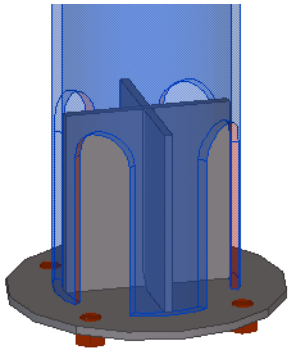
Box column base plate (1066) creates a base plate for a square or a circular column that is embedded in concrete.

Objects created

- Base plate
- Ribs
- Flow holes
- Bolts
- Welds

Use for

Situation	Description
	Base plate with a square column

Situation	Description
	Base plate with a square column
	Base plate with a circular column
	Base plate with a circular column

Before you start

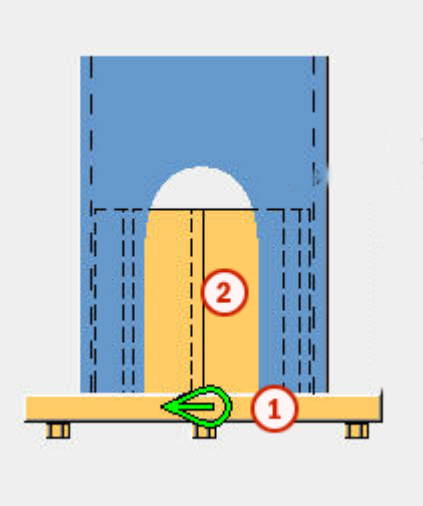
Create a column.

Selection order

1. Select the main part (column).
2. Pick a position.

   The detail is created automatically.

Part identification key

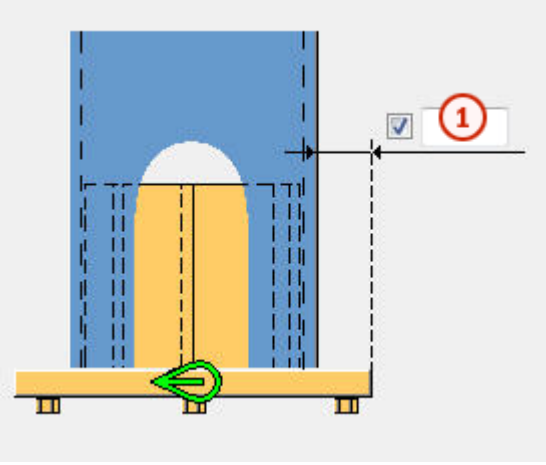


Number	Part
1	Base plate
2	Rib

Picture tab

Use the **Picture** tab to control the position of the base plate.

Dimensions



	Description	Default
1	Distance from the edge of the column to the edge of the base plate.	50 mm

Parts tab

Use the **Parts** tab to control the dimensions of the base plate, long rib, and short rib.

Plate

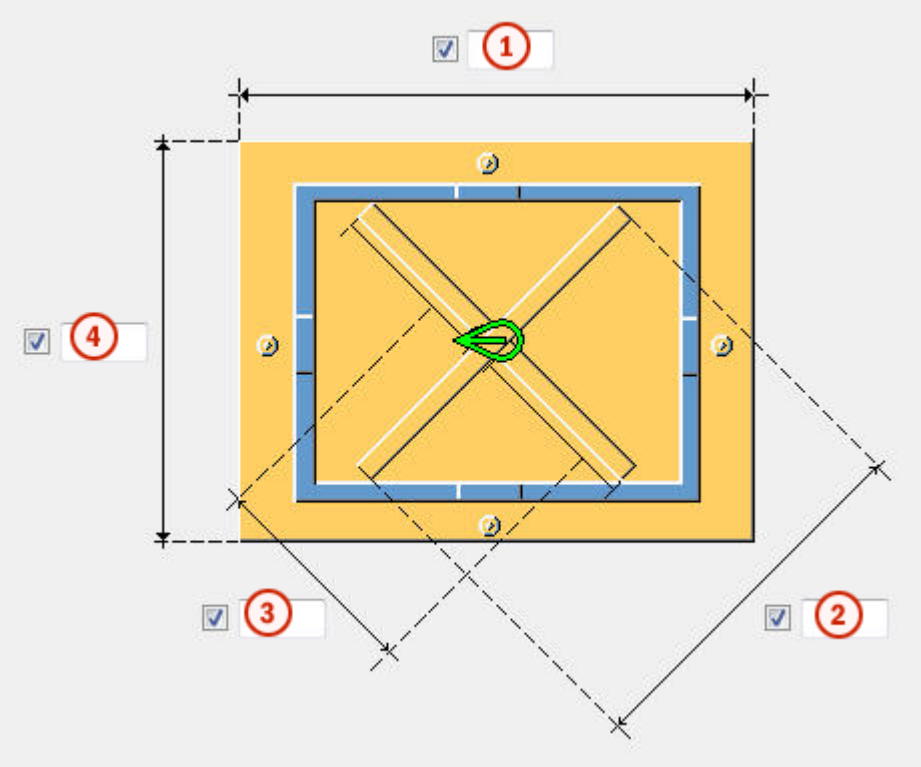
Option	Description	Default
Base plate	Base plate thickness.	
Long rib	Long rib thickness.	9 mm
Short rib	Short rib thickness.	9 mm

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tabs

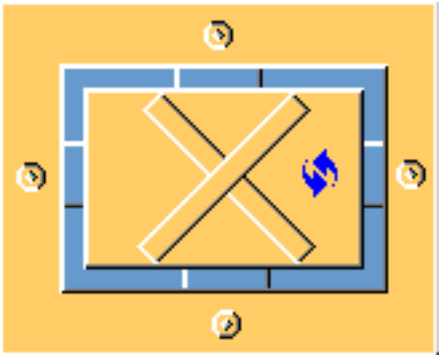
Use the **Parameters** tab to control the base plate and rib dimensions.

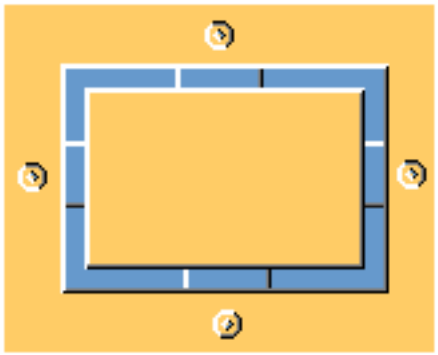
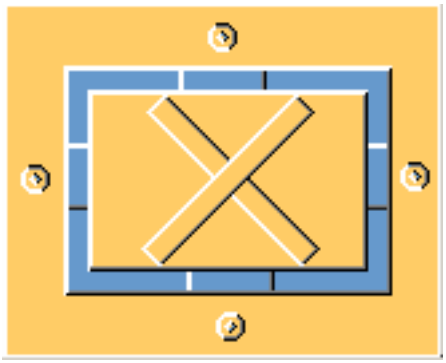
Base plate and rib dimensions



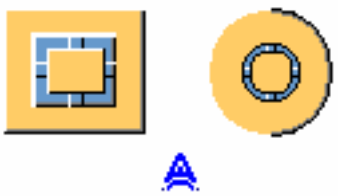
	Description
1	Width of base plate.
2	Length of long rib.
3	Length of short ribs.
4	Height of base plate.

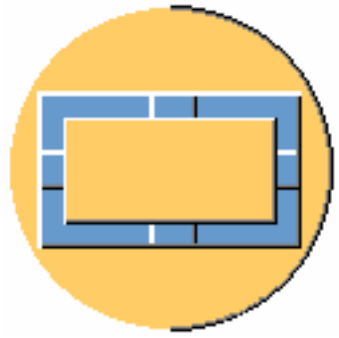
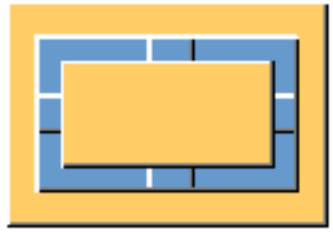
Rib creation

Option	Description
	Default No ribs are created. AutoDefaults can change this option.

Option	Description
	No ribs are created.
	Ribs are created.

Base plate creation

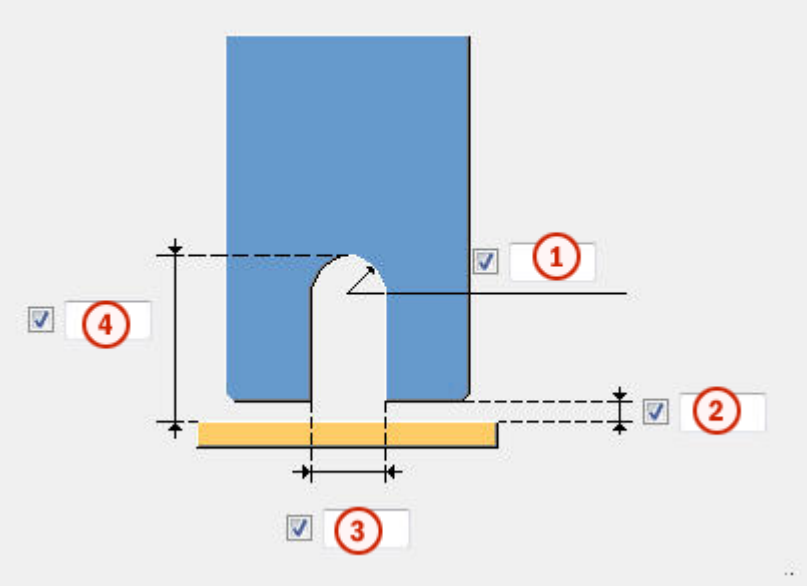
Option	Description
	Default Rectangular base plate AutoDefaults can change this option.
	Automatic

Option	Description
	Round base plate
	Rectangular base plate

Flow holes tab

Use the **Flow holes** tab to control the distance between the base plate and the column.

Flow hole properties



	Description
1	Rounding of the column flow hole.
2	Weld gap.

	Description
3	Width of the column flow hole.
4	Height of the column flow hole.

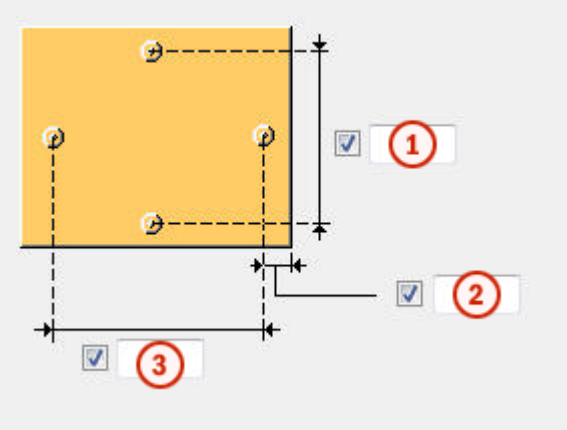
Flow hole creation

Option	Description
	Default Flow holes are created. AutoDefaults can change this option.
	Flow holes are not created.
	Flow holes are created.

Bolts tab

Use the **Bolts** tab to control the bolt properties.

Bolt positions



	Description
1	Vertical bolt spacing dimension.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Horizontal bolt spacing dimension.

Bolt basic properties

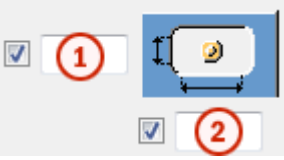
Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Slotted holes

You can define slotted, oversized, or tapped holes.

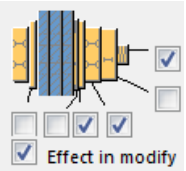


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Create as

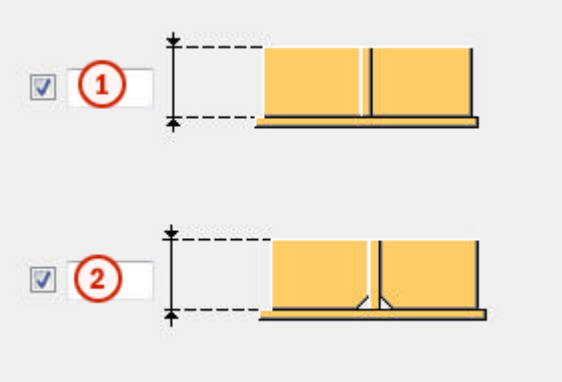
Use this option to switch between bolts and custom components.

Select the custom component from the **Applications & components** catalog and define the custom settings, up direction, rotation, and anchor length.

Height of ribs tab

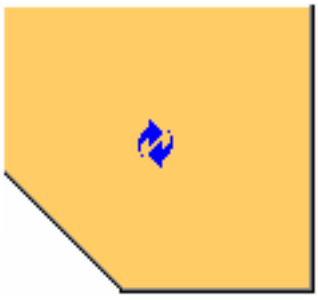
Use the **Height of ribs** tab to control the height of the short and the long ribs.

Height of ribs



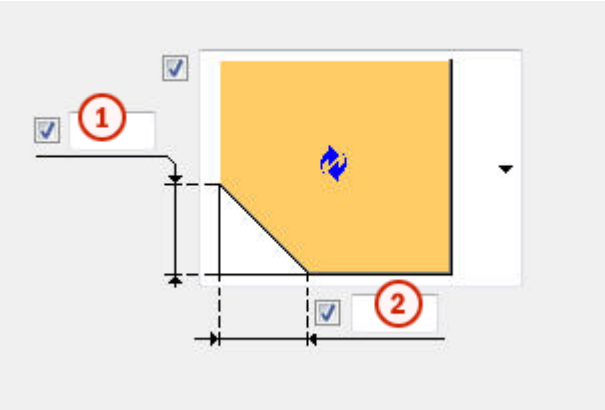
	Description	Default
1	Height of the long rib.	200 mm
2	Height of the short ribs.	200 mm

Short rib chamfer

Option	Description
	Default Line chamfer is created. AutoDefaults can change this option.

Option	Description
	Chamfer is not created.
	Line chamfer is created.
	Arc chamfer is created.

Chamfer dimensions



	Description	Default
1	Chamfer height.	15 mm
2	Chamfer width.	15 mm

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds tab

Click the link below to find out more:

5.16 Built-up Components

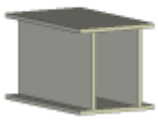
This section introduces built-up components available in Tekla Structures.

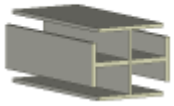
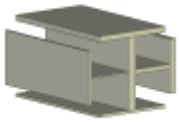
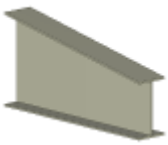
Click the links below to find out more:

- Beams (page 2745)
- Columns (page 2772)
- Frames (page 2784)
- Connections and details (page 2811)

Beams

Use these components to automatically create built-up beams:

Component	Image	Description
Box girder (S13) (page 2746)		Creates a built-up beam using four plates welded together.

Component	Image	Description
Cross profile (S32) (page 2750)		Creates a built-up beam using an I profile and two T profiles welded to the I profile web.
Cross plate profile (S33) (page 2755)		Creates a built-up beam from seven plates welded together.
Tapered beam (S98) (page 2759)		Creates a tapered or straight built-up beam with I shape.
Tapered beam 2 (S45) (page 2764)		Creates a tapered or straight built-up beam with I shape. Flange and web plates can be spliced.
PEB Tapered section (S94) (page 2769)		Creates a simple built-up beam or column between two picked points. The shape can be straight or tapered.

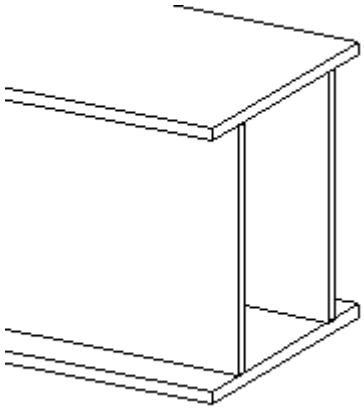
Box girder (S13)

Box girder (S13) creates a built-up beam with a box cross section.

Objects created

- Built-up beam
- Welds

Use for

Situation	Description
	Straight built-up beam with a cross section.

Do not use for

Tapered built-up beams.

Before you start

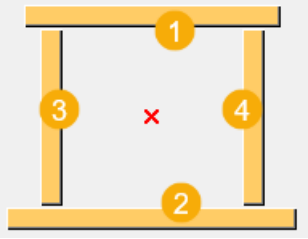
Ensure that you have two points to pick.

Selection order

1. Pick the start point for the beam.
2. Pick the end point for the beam.

   The beam is created automatically when you pick the end point.

Part identification key

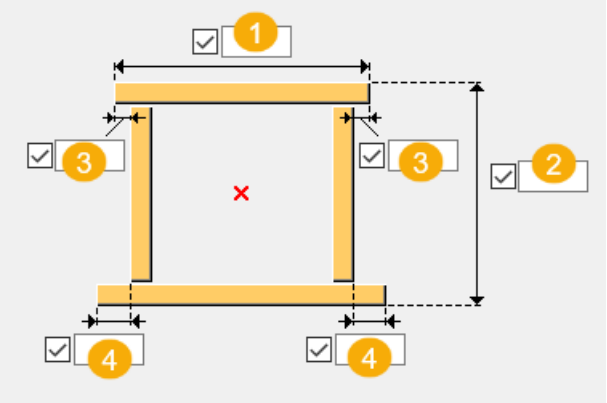


	Description
1	Top flange
2	Bottom flange
3	Left web
4	Right web

Picture tab

Use the **Picture** tab to define the dimensions.

Dimensions



	Description
1	Width of the beam
2	Height of the beam
3	Top flange extension dimension
4	Bottom flange extension dimension

Parts tab

Use the **Parts** tab to define the part properties.

Parts

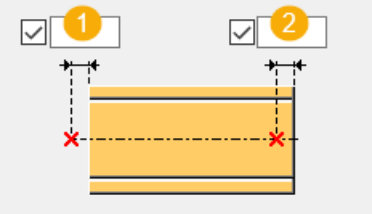
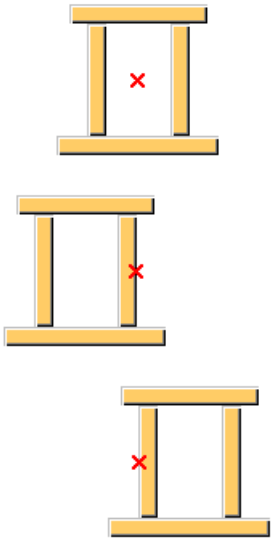
Option	Description
Top flange	Thickness of the top flange.
Bottom flange	Thickness of the bottom flange.
Left web	Thickness of the left web.
Right web	Thickness of the right web.

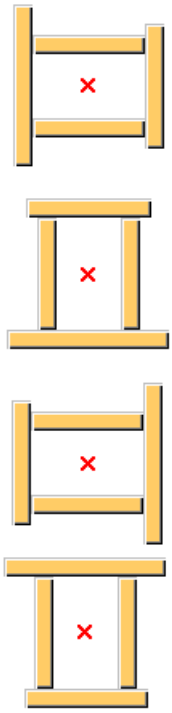
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in

Option	Description	Default
		File menu --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

Use the **Parameters** tab to define the part name, class and finish, and the position, rotation, and length.

Option	Description
Name	Define the name, class, and finish of the beam.
Class	
Finish	
	Negative values shorten the beam, positive values lengthen it. 1. Moves the beam end that you pick first. 2. Moves the beam end that you pick second.
	Select the position in plane and define the position in plane offset. See Part horizontal position (page 626).

Option	Description
	Select a suitable rotation and define the rotation offset. See Part rotation (page 622).
	Select the position in depth and define the position in depth offset. See Part position depth (page 623).

Welds

Click the link below to find out more:

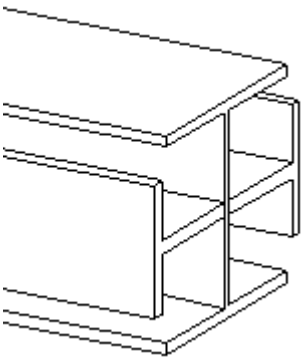
Cross profile (S32)

Cross profile (S32) creates a built-up beam using an I profile and two T profiles welded to the I profile web.

Objects created

- I profile
- T profile (2)
- Welds

Use for

Situation	Description
	Straight built-up beam. Two T profiles welded to an I profile.

Do not use for

Tapered built-up beams.

Before you start

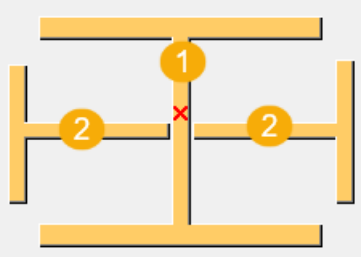
Check that you have two points to pick.

Selection order

1. Pick the start point for the beam.
2. Pick the end point for the beam.

   The beam is created automatically when you pick the end point.

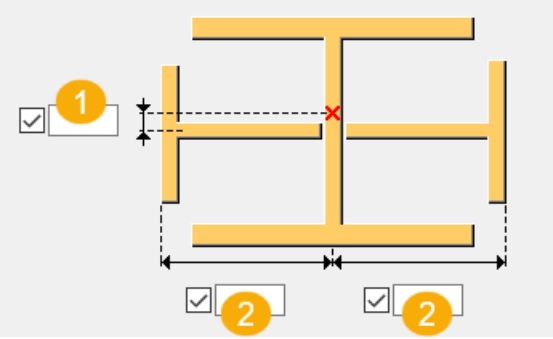
Part identification key



	Description
1	I profile
2	T profile

Picture tab
Use the **Picture** tab to define the dimensions of the profiles.

Dimensions



	Description
1	Adjust the location of the horizontal parts relative to the vertical parts. By default, the eccentricity is zero. T profiles are connected to the middle of the I profile web.
2	T profile dimension from the I profile center line.

Parts tab
Use the **Parts** tab to define the I profile and the T profile properties.

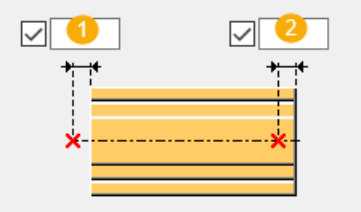
Parts

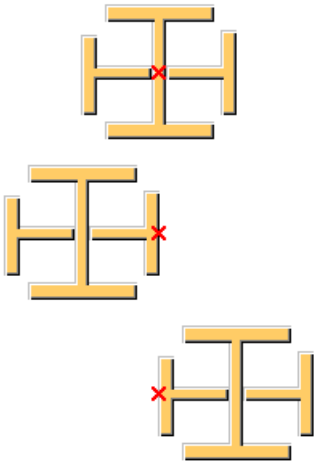
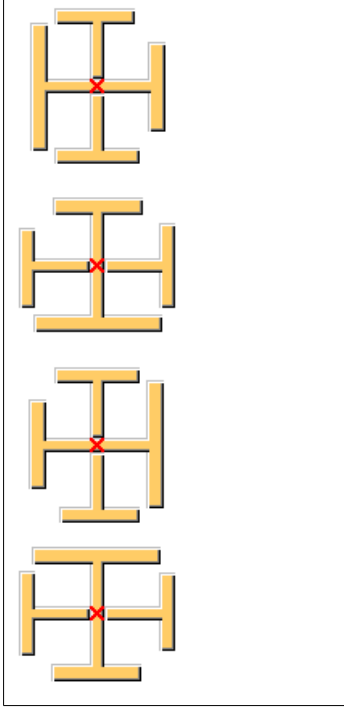
Option	Description
Main prof	Select the profile from the profile catalog.
Sec prof1	Select the profile from the profile catalog.
Sec prof2	Select the profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

Use the **Parameters** tab to define the part name, class and finish, and the position, rotation, and length.

Option	Description
Name	Define the name, class, and finish of the profiles.
Class	
Finish	
	Negative values shorten the beam, positive values lengthen it. 1. Moves the beam end that you pick first. 2. Moves the beam end that you pick second.

Option	Description
	Select the position in plane and define the position in plane offset. See Part horizontal position (page 626).
	Select a suitable rotation and define the rotation offset. See Part rotation (page 622).

Option	Description
	Select the position in depth and define the position in depth offset. See Part position depth (page 623).

Welds

Click the link below to find out more:

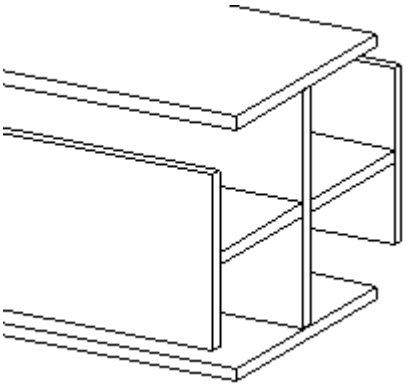
Cross plate profile (S33)

Cross plate profile (S33) creates a built-up beam using seven plates welded together.

Objects created

- Plates (7)

Use for

Situation	Description
	Straight built-up beam, seven plates welded together.

Do not use for

Tapered built-up beams.

Before you start

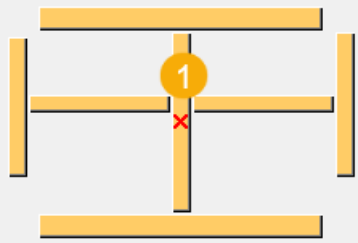
Ensure that you have two points to pick.

Selection order

1. Pick the start point for the beam.
2. Pick the end point for the beam.

   The beam is created automatically when you pick the end point.

Part identification key

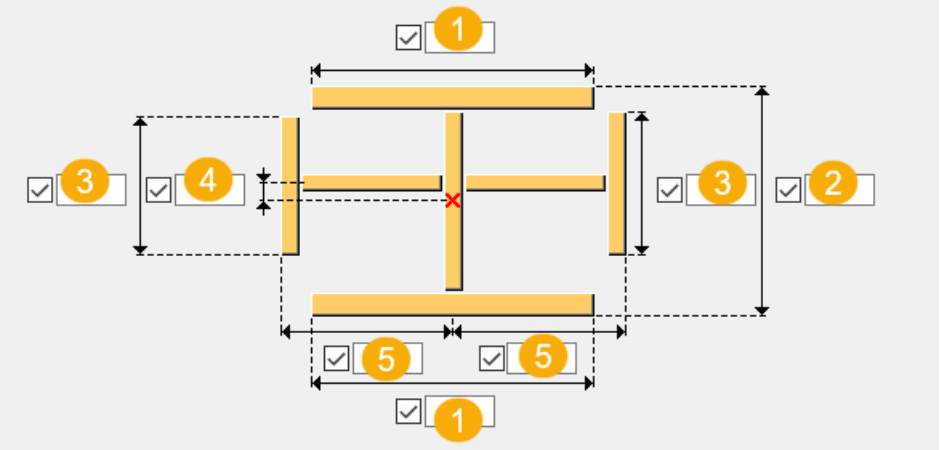


	Description
1	Cross plate. Seven plates are created.

Picture tab

Use the **Picture** tab define the dimensions.

Dimensions



	Description
1	Width of the beam
2	Height of the beam

	Description
3	Height of the plate
4	Adjust the location of the horizontal parts relative to the vertical parts. By default, the eccentricity is zero. Plates are connected to the middle of the vertical plate.
5	Plate edge dimension to the vertical plate center line.

Parts tab

Use the **Parts** tab to define the plate properties.

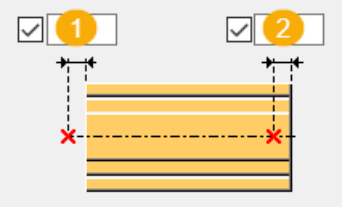
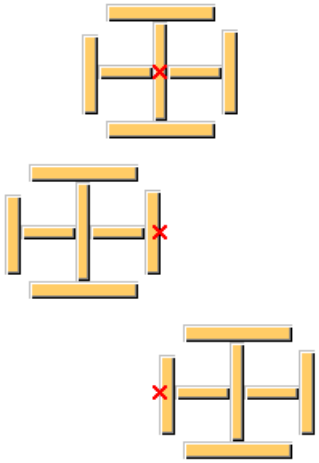
Parts

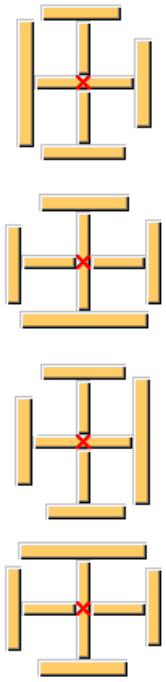
Option	Description
Top flange	Thickness of the top flange.
Bottom flange	Thickness of the bottom flange.
Vertical web	Thickness of the vertical web.
Left flange	Thickness of the left flange.
Right flange	Thickness of the right flange.
Left web	Thickness of the left web.
Right web	Thickness of the right web.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

Use the **Parameters** tab to define the part name, class and finish, and the position, rotation, and length.

Option	Description
Name	Define the name, class, and finish of the plates.
Class	
Finish	
	Negative values shorten the beam, positive values lengthen it. 1. Moves the beam end that you pick first. 2. Moves the beam end that you pick second.
	Select the position in plane and define the position in plane offset. See Part horizontal position (page 626).

Option	Description
	Select a suitable rotation and define the rotation offset. See Part rotation (page 622).
	Select the position in depth and define the position in depth offset. See Part position depth (page 623).

Welds

Click the link below to find out more:

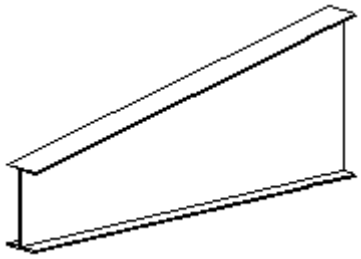
Tapered beam (S98)

Tapered beam (S98) creates a tapered or straight built-up beam with I shape.

Objects created

- Web plate
- Top flange plate
- Bottom flange plate
- Welds

Use for

Situation	Description
	Creates a simple built-up beam between two picked points.

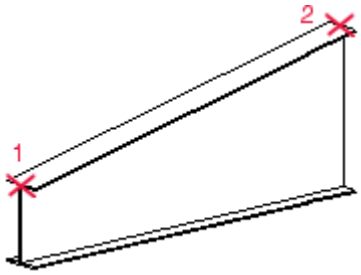
Before you start

Ensure that you have two points to pick.

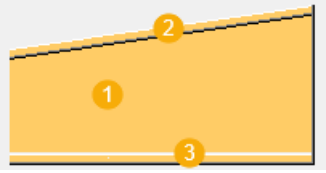
Selection order

1. Pick the start point of the beam.
2. Pick the end point of the beam.

   The beam is created automatically when you pick the end point.



Part identification key



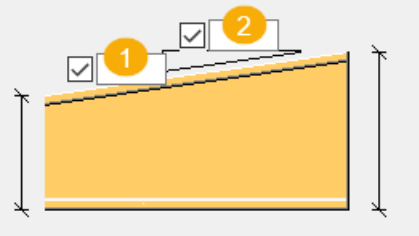
	Description
1	Web plate

	Description
2	Top flange plate
3	Bottom flange plate

Picture tab
Use the **Picture** tab to define the height of the web or the entire beam, sloping rise, and run values.

Dimensions

Select the beam height type on the **Parameters** tab.



	Description
1	Length of the slope
2	Height of the slope

Parts tab
Use the **Parts** tab to define the properties of the plates.

Parts

Option	Description
Web plate	Thickness and width of the web plate.
Top fl. profile	Select the profile from the profile catalog.
Bottom fl. profile	Select the profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

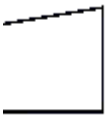
Option	Description	Default
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

Use the **Parameters** tab to define the beam end shape, height type, type of reference point, beam end setback distance, and the position of the web plate.

Type of cut at beam end

Select the shape of the beam end at the beginning and end of the beam.

Option	Description
Vertical in global system	
Perpendicular to top flange	
In Z of current plane	Depends on the position of the work plane.

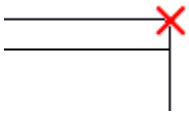
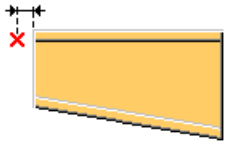
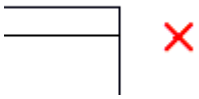
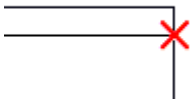
Beam height type

Affects the height values entered on the **Picture** tab.

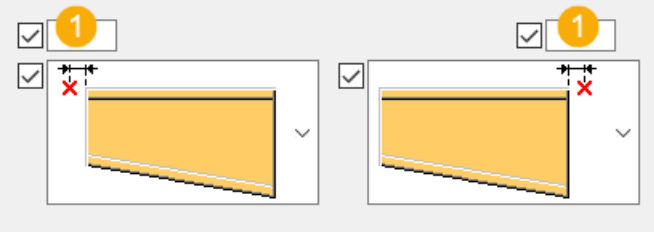
Option	Description
	Height of the web.
	Height of the entire beam.

Type of reference point

Select the location of the end of the beam, relative to the point you pick.

Option	Option	Description
		Top profile, setback distance.
		Top profile, end of the web plate. 
		Top web plate, setback distance. 
		Top web plate, end of the web plate. 

Beam end setback distance



	Description
1	Define the the setback distance of beam ends.

Position of web plate

Option	Description
	Select the location of the web plate. See also Part position depth (page 623).

Welds

Click the link below to find out more:

Tapered beam 2 (S45)

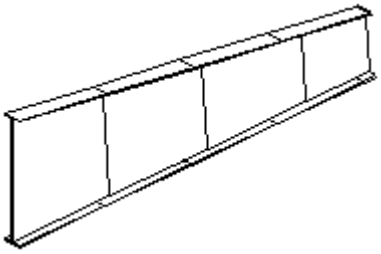
Tapered beam 2 (S45) creates a built-up beam with I shape. The beam can be tapered or straight. You can control the size of the spliced material.

Objects created

- Web plates
- Top flange plates
- Bottom flange plates

Use for

Situation	Description
	A simple, straight beam built up from two flange plates and a web plate.

Situation	Description
	A tapered beam built-up from several spliced plates. The points you pick determine the beam length. You can adjust the length on the **Picture** tab.

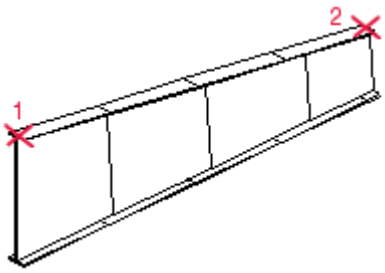
Before you start

Ensure that you have two points to pick.

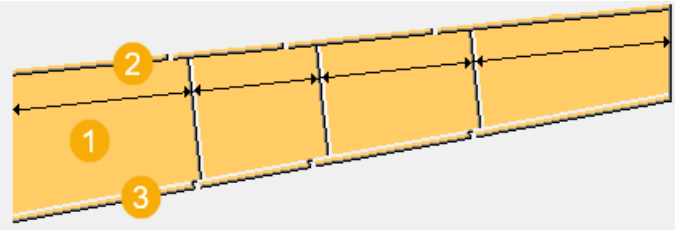
Selection order

1. Pick the start point for the beam.
2. Pick the end point for the beam.

   The beam is created automatically when you pick the end point.



Part identification key

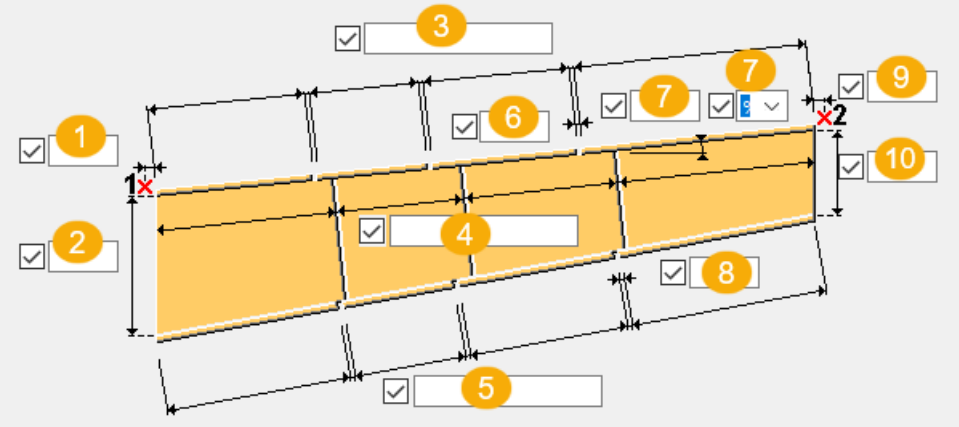


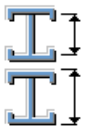
	Description
1	Web plate
2	Top flange plate
3	Bottom flange plate

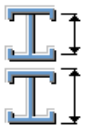
Picture tab

Use the **Picture** tab to define the beam dimensions and location, relative to the points picked.

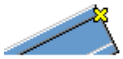
Dimensions



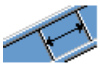
	Description
1	Distance from the first point picked to the end of the beam. A negative value lengthens the beam, a positive value shortens it.
2	Beam depth at the first end. This is either the height of the web or the entire beam, depending on what you select as the depth measure. 
3	Length of the top flange plates. For example, to have four one-meter sections, enter `4*1000`. If you want to create the flange or web from a single plate, leave the box empty.
4	Length of the web plates. For example, to have four one-meter sections, enter `4*1000`. If you want to create the flange or web from a single plate, leave the box empty.
5	Length of the bottom flange plates. For example, to have four one-meter sections, enter `4*1000`. If you want to create the flange or web from a single plate, leave the box empty.
6	Gap between the top flange plates.
7	Horizontal sloping of the beam, either as a percentage or in degrees.
8	Gap between the bottom flange plates.

	Description
9	Distance from the last point picked to the end of the beam. A negative value lengthens the beam, a positive value shortens it.
10	Beam depth at the second end. This is either the height of the web or the entire beam, depending on what you select as the depth measure. 

Beam position to picked point

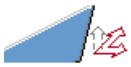
Option	Option	Description
		Moves the beam so that the point is located at the top of the top flange.
		Moves the beam so that the point is located at the bottom of the top flange.
		Moves the beam so that the point is located in the middle of the beam cross section.

Web plate orientation

Option	Description
	Web plates are cut perpendicular to the top flange.
	Web plates are cut vertically.

Beam end alignment

Option	Option	Description
		Cut is vertical or horizontal.
		Cut is perpendicular to the top flange.

Option	Option	Description
		Cut is relative to the current position of the work plane.

Depth measure

Option	Description
	Depth is calculated from the outer surfaces of the top and bottom flanges.
	Depth is the depth of the web.

Parts tab

Use the **Parts** tab to define the plate properties.

Parts

Option	Description
Top fl. profile	Select the profile from the profile catalog.
Bottom fl. profile	Select the profile from the profile catalog.
Web pl. thickness	Thickness of the web plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Option	Description	Default
Finish	Describes how the part surface has been treated.	

Welds

Click the link below to find out more:

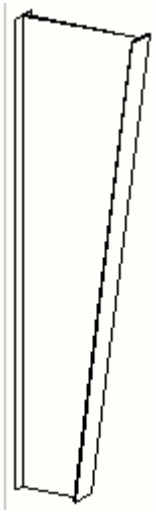
PEB Tapered section (S94)

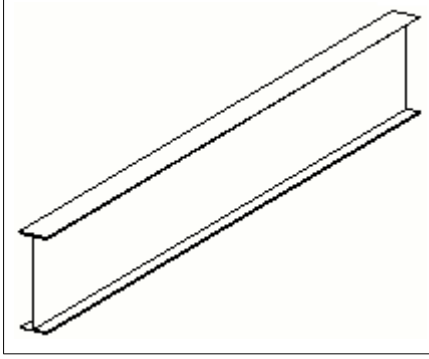
PEB Tapered section (S94) creates a simple built-up beam or column between two picked points. The shape can be straight or tapered.

Objects created

- Web plate
- Flange plates (2)
- Welds

Use for

Situation	Description
	Tapered built-up column

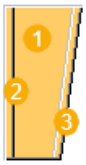
Situation	Description
	Straight built-up beam

Selection order

1. Pick the start point for the beam or column.
2. Pick the end point for the beam or colum.

   The beam or column is created automatically when you pick the end point.

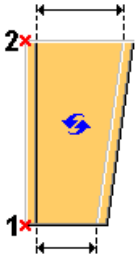
Part identification key

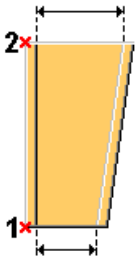
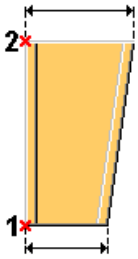


	Description
1	Web plate
2	Outer flange plate
3	Inner flange plate

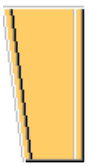
Picture tab
Use the **Picture** tab to define the dimensions and the rotation.

Dimensions

Option	Description
	Default Define the column web dimensions at the top and the bottom. AutoDefaults can change this option.

Option	Description
	Define the column web dimensions at the top and the bottom.
	Define the column flange dimensions at the top and the bottom.

Rotation

Option	Description
	Default Column is not rotated. AutoDefaults can change this option.
	Column is rotated.
	Column is not rotated.

Parts tab

Use the **Parts** tab to define the part properties.

Parts

Option	Description
Web thickness	Thickness of the web plate.

Option	Description
Outer flange	Thickness and width of the outer flange plate.
Inner flange	Thickness and width of the inner flange plate.

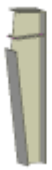
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Welds

Click the link below to find out more:

Columns

Use these components to automatically create built-up columns:

Component	Image	Description
Tapered column (S99) (page 2773)		Creates a single built-up column, which can be straight or tapered.
Tapered column 2 (S44) (page 2778)		Creates a built-up column, which can be straight or tapered. Flange and web plates can be spliced.

Component	Image	Description
PEB Tapered section (S94) (page 2769)		Creates a simple built-up beam or column between two picked points. The shape can be straight or tapered.

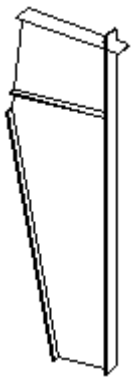
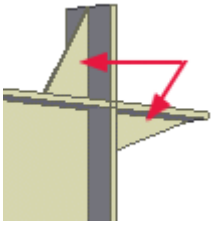
Tapered column (S99)

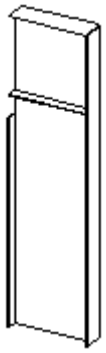
Tapered column (S99) creates a single built-up column, which can be straight or tapered.

Objects created

- Vertical web plate
- Vertical outer flange plate
- Inclined inner flange plate
- Top plate
- Flange stiffeners (2)
- Horizontal stiffeners (2)
- Welds

Use for

Situation	Description
	A tapered built-up column with a stiffened corner. Flange stiffeners: 

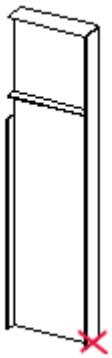
Situation	Description
	A straight built-up column.

Before you start

Ensure that you have a point to pick.

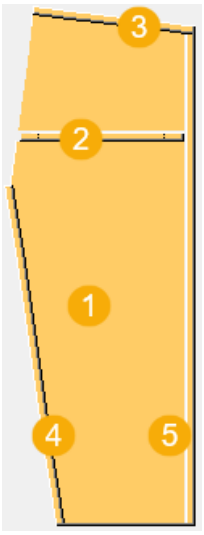
Selection order

1. Pick the position of the column:



The column is created automatically when you pick the position.

Part identification key

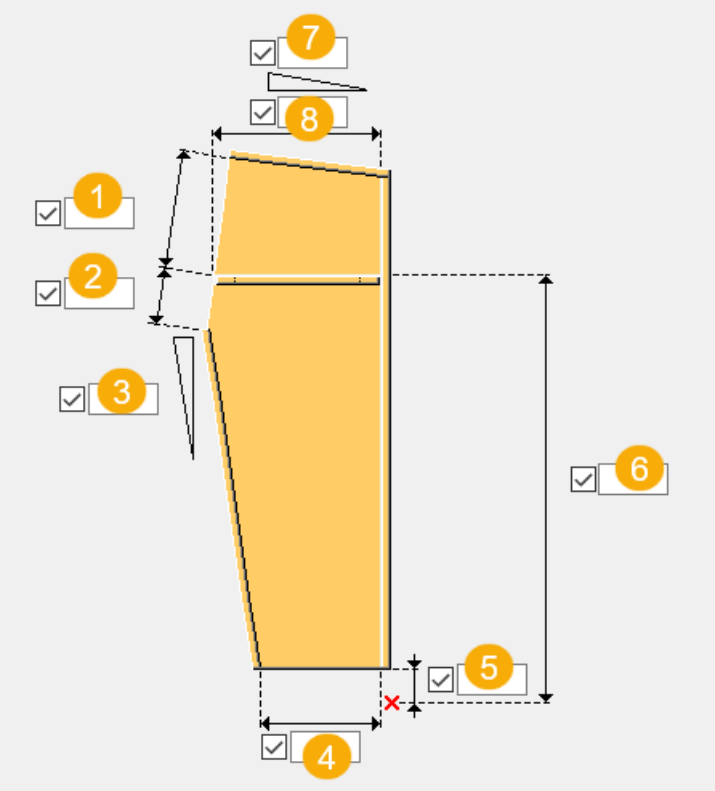


	Description
1	Web plate
2	Horizontal stiffener
3	Top plate
4	Inner flange plate
5	Outer flange plate

Picture tab

Use the **Picture** tab to define the column dimensions.

Dimensions



	Description
1	Dimension from the web plate top corner to the top of the stiffener
2	Dimension from the top of the stiffener to the inner corner of the web plate
3	Inner flange plate slope
4	Web plate width at the bottom
5	Web plate vertical offset from the input point

	Description
6	Web plate dimension from the input point to the top of the horizontal stiffener
7	Top plate slope as a percentage, for example, 10.
8	Horizontal stiffener length

Parts tab
Use the **Parts** tab to define the plate properties.

Parts

Option	Description
Web	Thickness of the web
Outer flange, **Inner flange**, **Top plate**, **Horiz stiffener**	Select the profiles on the **Parameters** tab.
Flange stiffener	Thickness of the flange stiffener

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab
Use the **Parameters** tab to define the plate profiles, orientation, and dimensions.

Profiles

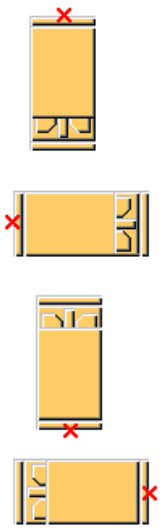
Option	Description
Outer flange profile, **Inner flange profile**, **Top plate profile**, **Horizontal stiffener profile**	Profile size of the plates. The `std_flange_plates.dat` file defines the available plate profiles.

Flange splice dimension

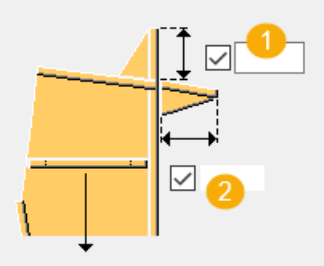
If the plate is longer than the maximum plate length defined in the `std_flange_plates.dat` file, Tekla Structures creates a splice.

	Description
1	Inner flange splice dimension
2	Outer flange splice dimension

Orientation

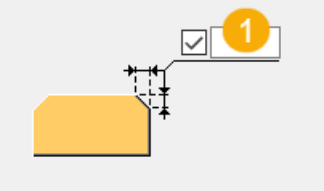
Option	Description
	Select a suitable rotation. The column web is rotated relative to the work plane.

Extra length



	Description
1	Outer flange extra length
2	Top plate extra length

Horizontal stiffener chamfer



	Description
1	Internal chamfer dimension in the horizontal stiffener. The default value is `15`.

Welds

Click the link below to find out more:

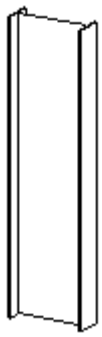
Tapered column 2 (S44)

Tapered column 2 (S44) creates a welded built-up column, which can be tapered or straight. You can control the taper and the size of the spliced material.

Objects created

- Web plates
- Top flange plates
- Bottom flange plates
- Welds

Use for

Situation	Description
	A simple straight column built up from two flange plates and a web plate.
	A more complex tapered column, with several plates forming the web, and top and bottom flanges.

Before you start

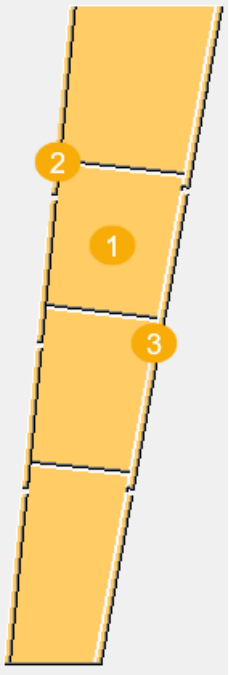
Ensure that you have a point to pick.

Selection order

1. Pick the position of the column.

   The column is created automatically when you pick the position.

Part identification key

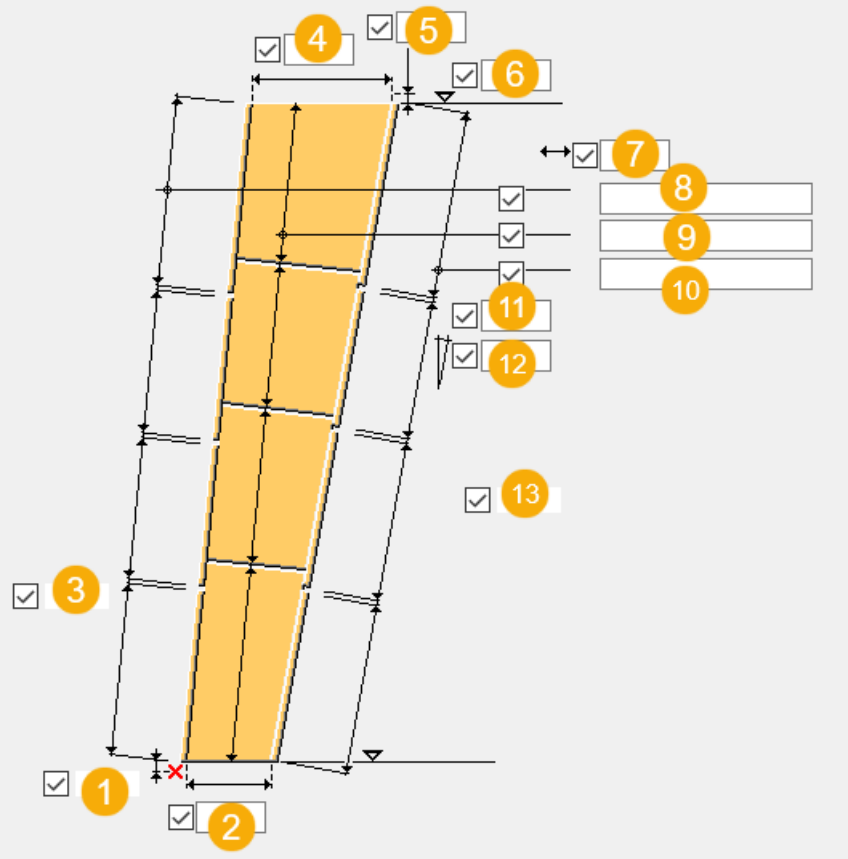


	Description
1	Web plate
2	Top flange plate
3	Bottom flange plate

Picture tab

Use the **Picture** tab to define the column dimensions and location relative to the picked points.

Dimensions



	Description
1	Distance from the first point picked to the end of the column.
2	Column width at the first end.
3	Gap between the top flange plates.
4	Column width at the second end.
5	Distance from the last point picked to the end of the column.
6	Column height.
7	Column end alignment dimension.
8	Length of the top flange plates. For example, to have four one-meter sections, enter `4*1000`. If you want to create the flange or web from a single plate, leave the box empty.
9	Length of the web plates. For example, to have four one-meter sections, enter `4*1000`. If you want to create the flange or web from a single plate, leave the box empty.

	Description
10	Length of the bottom flange plates. For example, to have four one-meter sections, enter `4*1000`. If you want to create the flange or web from a single plate, leave the box empty.
11	Gap between the bottom flange plates.
12	Horizontal sloping of the beam, as a percentage.
13	Column rotation dimension.

Column rotation

Option	Description
	Select how the column is rotated and define the dimension for the rotation.

Column position to picked point

Option	Description
	Moves the column so that the point is located at the column flange.
	Moves the column so that the point is located at the column web.
	Moves the column so that the point is located in the middle of the column cross section.
	Moves the column so that the point is located at the column web.
	Moves the column so that the point is located at the column flange.

Web plate orientation

Option	Description
	Web plates are cut perpendicular to the top flange.
	Web plates are cut vertically.

Column end alignment

Option	Description
	Cut is vertical or horizontal.
	Cut is perpendicular to the top flange.
	Cut is relative to the current position of the work plane.

Depth measure

Option	Description
	Depth is calculated from the outer surfaces of the top and bottom flanges.
	Depth is the depth of the web.

Column flange

Option	Description
	Column flange is perpendicularly fitted to the web.
	Column flange is aligned with the web.

Parts tab

Use the **Parts** tab to define the plate properties.

Parts

Option	Description
Top fl. profile	Select the profile from the profile catalog.
Bottom fl. profile	Select the profile from the profile catalog.
Web pl. thickness	Thickness of the web plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Welds

Click the link below to find out more:

Frames

Use the following components to automatically create built-up frames, beams, and columns:

Component	Image	Description
Tapered frame comonor (S53) (page 2785)		Creates a built-up frame or parts of it. You can control the taper and the size of the spliced material.
PEB Frame (S92) (page 2806)		Creates a pre-engineered building frame to the selected point. This component uses existing components to create built-up columns and

Component	Image	Description
		rafters, base plates, knee connection, splice connections, and apex connection.

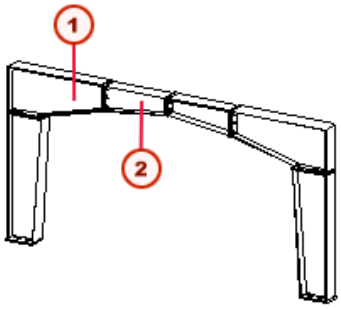
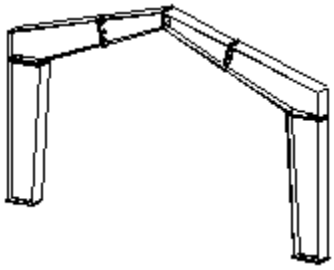
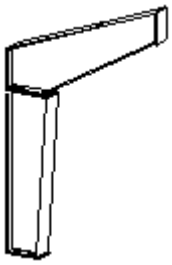
Tapered frame comonor (S53)

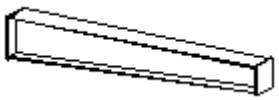
Tapered frame comonor (S53) creates a built-up frame or parts of it. You can control the taper and the size of the spliced material.

Objects created

- Built-up columns with base plate and end plate
- Built-up beams with end plates

Use for

Situation	Description
	Built-up frame **1** Beam 1 **2** Beam 2 On the **Picture** tab, use the following options: • Frame type: Column, first beam, and second beam are created. • Symmetry type: Symmetrical frame is created.
	Sloped built-up frame On the **Picture** tab, use the following options: • Frame type: Column, first beam, and second beam are created. • Symmetry type: Symmetrical frame is created.
	Only half of the frame On the **Picture** tab, use the following options: • Frame type: Column and first beam are created. • Symmetry type: Symmetrical frame is not created.

Situation	Description
	Built-up column with base plate and end plate On the **Picture** tab, use the following options: • Frame type: Column is created. • Symmetry type: Symmetrical frame is not created.
	Built-up beam (1) with end plates On the **Picture** tab, use the following options: • Frame type: First beam is created. • Symmetry type: Symmetrical frame is not created.
	Built-up beam (2) with end plates On the **Picture** tab, use the following options: • Frame type: Second beam is created. • Symmetry type: Symmetrical frame is not created.

Limitations

Creates a frame only in the global X direction. Y direction is not possible.

Before you start

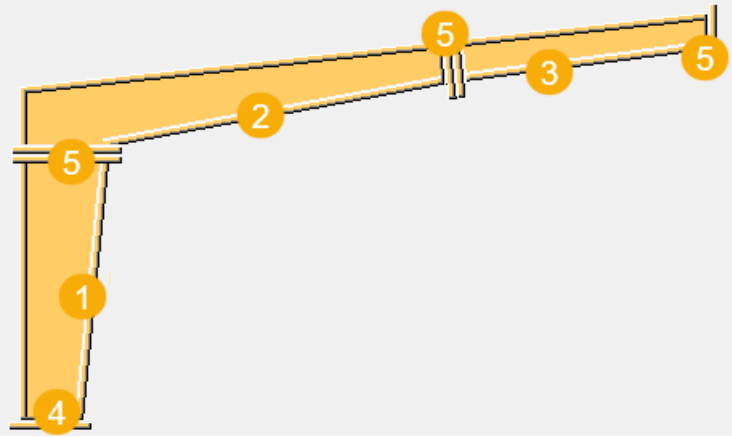
Ensure that you have a point to pick.

Selection order

1. Pick the position of the column.

   The frame is created automatically when you pick the position.

Part identification key

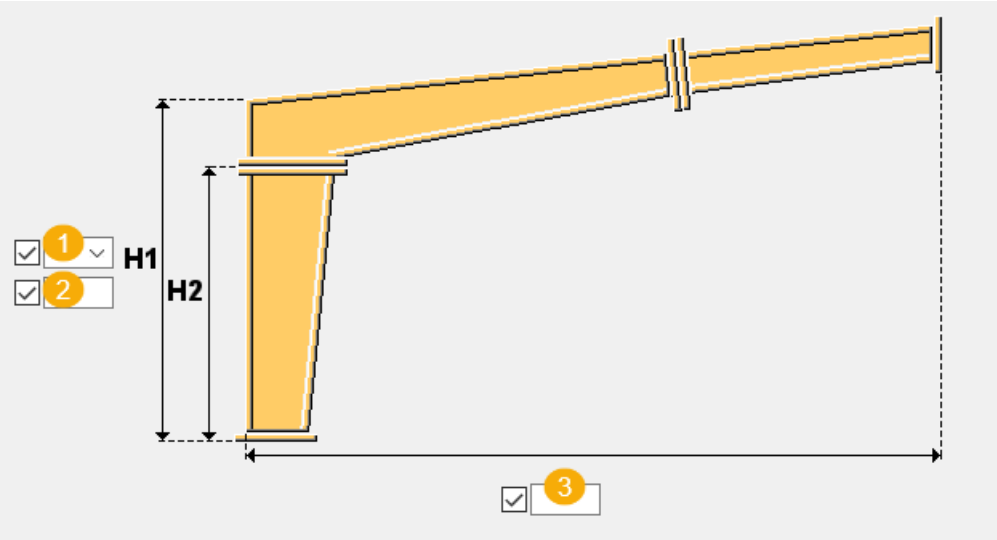


	Description
1	Column
2	First beam
3	Second beam
4	Base plate
5	End plate

Picture tab

Use the **Picture** tab to define the frame dimensions and location relative to the picked points.

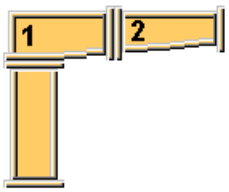
Dimensions



	Description
1	Select the height dimension to be defined.

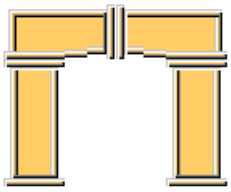
	Description
2	Define the height.
3	Dimension from the column outer edge to the frame center line.

Frame type

Option	Description
	Column is created.
	First beam is created.
	Second beam is created.
	Column and first beam are created.
	Column, first beam, and second beam are created.

Symmetry type

Option	Description
	Symmetrical frame is not created.

Option	Description
	Symmetrical frame is created.

Part class

Define the class of the web and flange plates, base plates, and end plates.

Parts tab

Use the **Parts** tab to define the plate properties.

Parts

Column parts	Description
Top fl. profile, **Bottom fl. profile**	Select the profile from the profile catalog.
Web thickness	Thickness of the column web.
Base plate	Thickness and width of the base plate.
Horiz end plate	Thickness and width of the horizontal end plate.

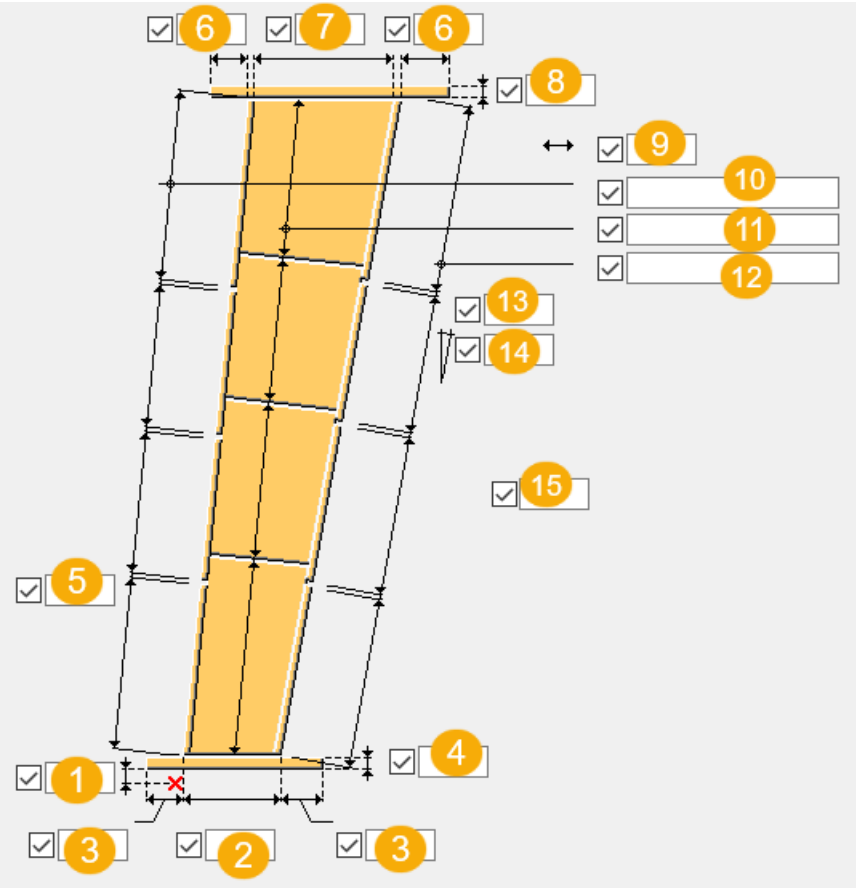
1 Beam parts	Description
Top fl. profile, **Bottom fl. profile**, **Vertical fl. profile**	Select the profile from the profile catalog.
Web thickness	Thickness of the first beam web.
Horiz end plate	Thickness and width of the horizontal end plate.
Lateral end plate	Thickness and width of the lateral end plate.

2 Beam parts	Description
Top fl. profile, **Bottom fl. profile**	Select the profile from the profile catalog.
Web thickness	Thickness of the second beam web.
Left end plate	Thickness and width of the left end plate.
Right end plate	Thickness and width of the right end plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Column tab
Use the **Column** tab to define the column dimensions.

Dimensions



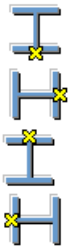
	Description
1	Distance from the first point picked to the end of the column.
2	Column depth at the first end. This is either the height of the web or the entire column, depending on what you select as the depth measure.
3	Base plate extension dimension at the bottom of the column.
4	Base plate thickness.
5	Gap between the top flange plates.
6	End plate extension dimension at the top of the column.
7	Column depth at the second end. This is either the height of the web or the entire column, depending on what you select as the depth measure.
8	End plate thickness.
9	Column end alignment dimension.

	Description
10	Length of the top flange plates. For example, to have four one-meter sections, enter `4*1000`. If you want to create the flange from a single plate, leave the box empty.
11	Length of the web plates. For example, to have four one-meter sections, enter `4*1000`. If you want to create the web from a single plate, leave the box empty.
12	Length of the bottom flange plates. For example, to have four one-meter sections, enter `4*1000`. If you want to create the flange from a single plate, leave the box empty.
13	Gap between the bottom flange plates.
14	Sloping of the column as a percentage.
15	Column rotation dimension.

Column end alignment

Option	Description
	Cut is vertical or horizontal.
	Cut is perpendicular to the top flange.
	Cut is relative to the current position of the work plane.

Column rotation

Option	Description
	Select how the column is rotated and define the dimension for the rotation.

Column position to picked point

Option	Description
	Moves the column so that the point is located at the column flange.
	Moves the column so that the point is located at the column web.
	Moves the column so that the point is located in the middle of the column cross section.

Web plate orientation

Option	Description
	Web plates are cut perpendicular to the top flange.
	Web plates are cut vertically.

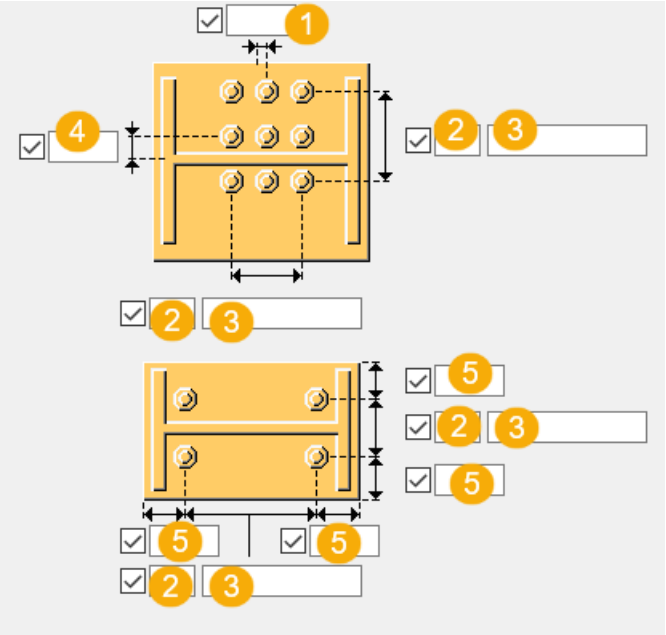
Depth measure

Option	Description
	Depth is the depth of the web.
	Depth is calculated from the outer surfaces of the top and bottom flanges.

Column 2 tab

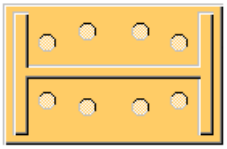
Use the **Column 2** tab to define the base plate bolt group dimensions and properties.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Dimension for vertical bolt group position.
5	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.

Bolt creation

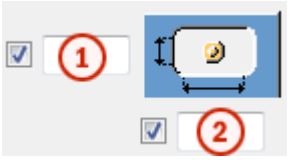
Option	Description
	Holes are created.
	Bolts are created.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



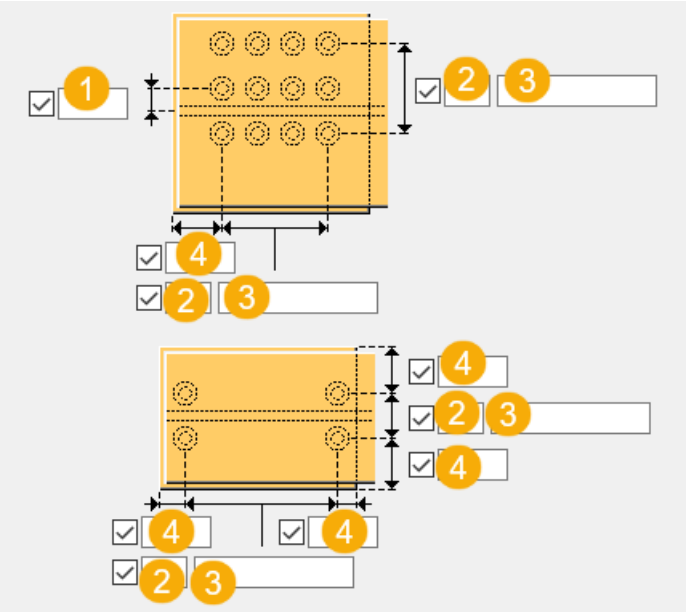
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	

Option	Description	Default
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Column 3 tab

Use the **Column 3** tab to define the end plate bolt group dimensions and properties between the column and the first beam.

Bolt group dimensions



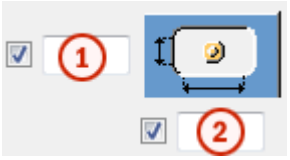
	Description
1	Dimension for vertical bolt group position.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



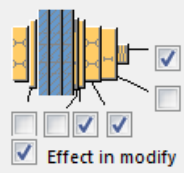
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	

Option	Description	Default
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

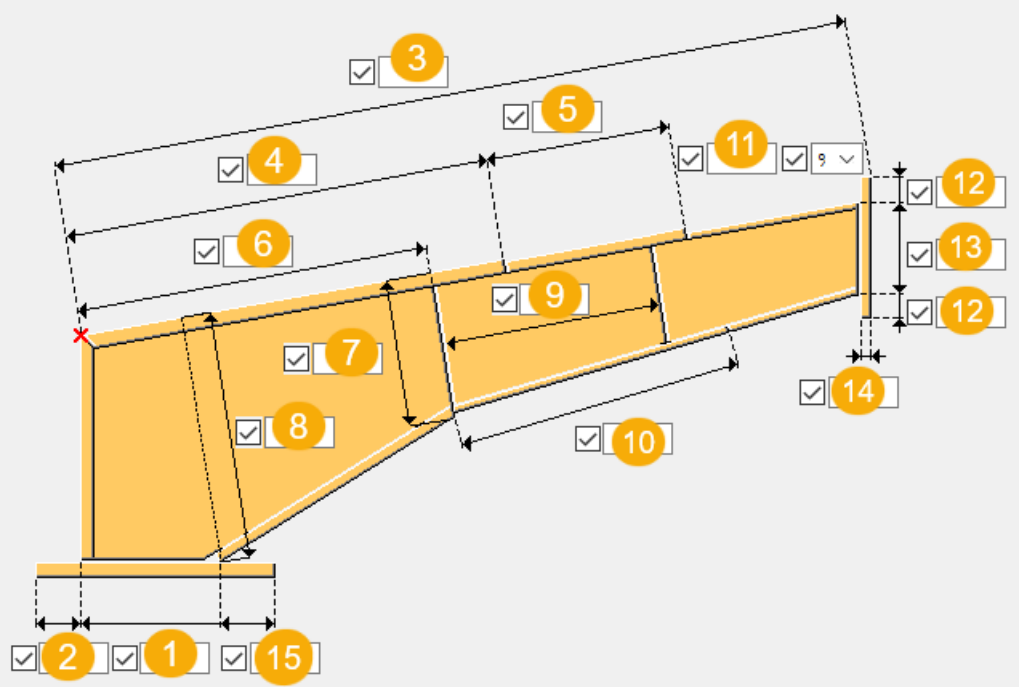
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Beam 1 tab

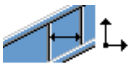
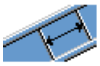
Use the **Beam 1** tab to define the dimensions of the first beam.

Dimensions



	Description
1	Beam depth at the first end.
2	Distance from the first point picked to the end of the beam.
3	Length of the top flange.
4	Length of the top flange plate to the first flange gap.
5	Length of the top flange plate to the second flange gap.
6	Length the top flange to the slope angle.
7	Beam web height at the slope angle.
8	Beam web height at the column end.
9	Length of the web plate.
10	Length of the bottom flange plate.
11	Horizontal sloping of the beam, either as a percentage or in degrees.
12	End plate extension dimension.
13	Beam depth at the second end.
14	End plate thickness.
15	End plate extension dimension.

Web plate orientation

Option	Description
	Web plates are cut perpendicular to the top flange.
	Web plates are cut vertically.

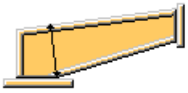
Beam end alignment

Option	Description
	Cut is vertical or horizontal.
	Cut is perpendicular to the top flange.

Beam length

Option	Description
	Sloped beam length
	Horizontal beam length

Beam height

Option	Description
	Perpendicular beam height
	Vertical beam height

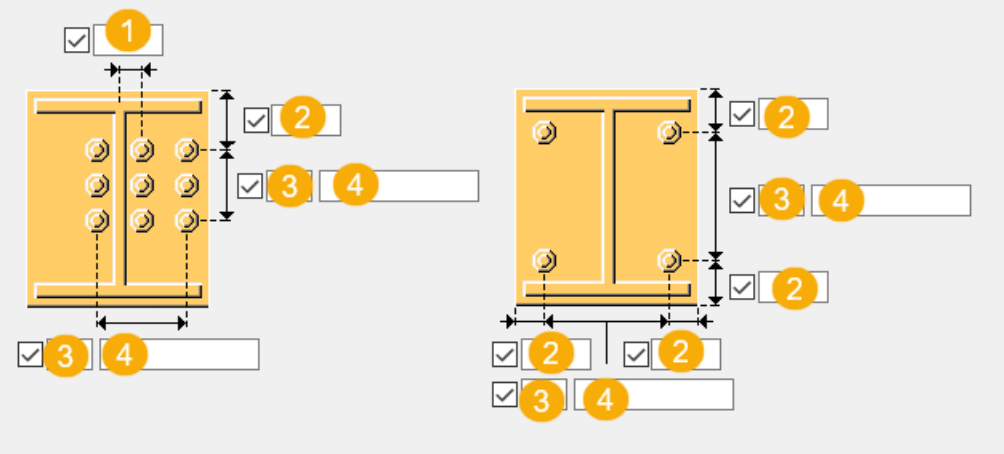
Beam bottom cut

Option	Description
	Beam bottom web is cut.
	Beam bottom is fitted to the end plate.

Beam 1_2 tab

Use the **Beam 1_2** tab to define the end plate bolt group dimensions and bolt properties between the first beam and the second beam.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

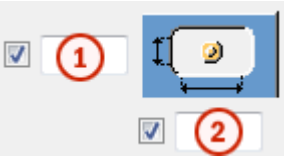
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

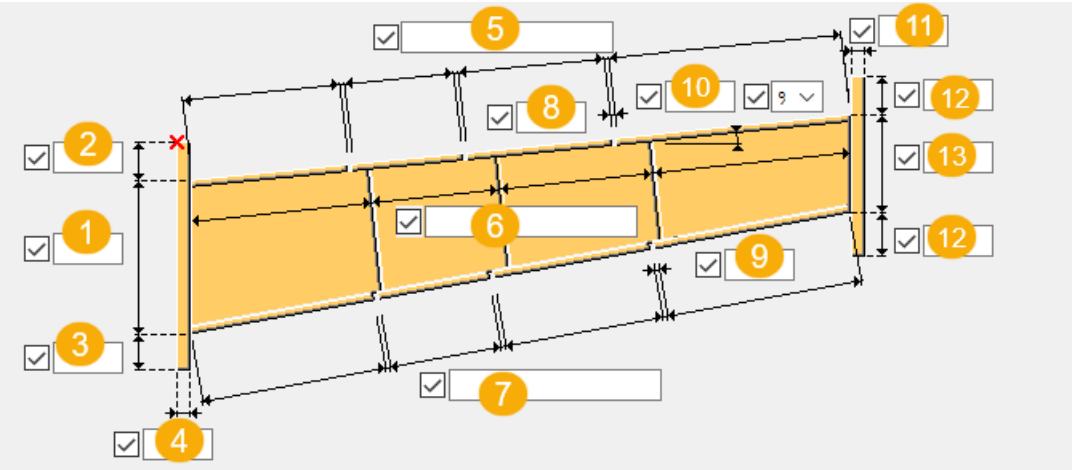


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Beam 2 tab

Use the **Beam 2** tab to define the dimensions of the second beam.

Dimensions



	Description
1	Beam depth at the first end.
2	Distance from the first point picked to the end of the beam.
3	End plate extension at the first end.
4	End plate thickness at the first end.
5	Length of the top flange plates. For example, to have four one-meter sections, enter `4*1000`. Leave blank to create the flange or web from a single plate.
6	Length of the web plates. For example, to have four one-meter sections, enter `4*1000`. Leave blank to create the flange or web from a single plate.
7	Length of the bottom flange plates. For example, to have four one-meter sections, enter `4*1000`. Leave blank to create the flange or web from a single plate.
8	Gap between the top flange plates.
9	Gap between the bottom flange plates.
10	Horizontal sloping of the beam, either as a percentage or in degrees.
11	End plate thickness at the second end.
12	End plate extension at the second end.
13	Beam depth at the second end. This is the height of the web or the entire beam, depending on what you select as the depth measure.

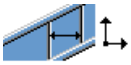
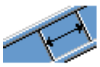
Beam position to picked point

Option	Option	Description
		Moves the beam so that the point is located at the beam flange.
		Moves the beam so that the point is located at the beam web.
		Moves the beam so that the point is located in the middle of the beam cross section.

Beam end alignment

Option	Option	Option
		Cut is vertical or horizontal.
		Cut is perpendicular to the top flange.
		Cut is relative to the current position of the work plane.

Web plate orientation

Option	Description
	Web plates are cut perpendicular to the top flange.
	Web plates are cut vertically.

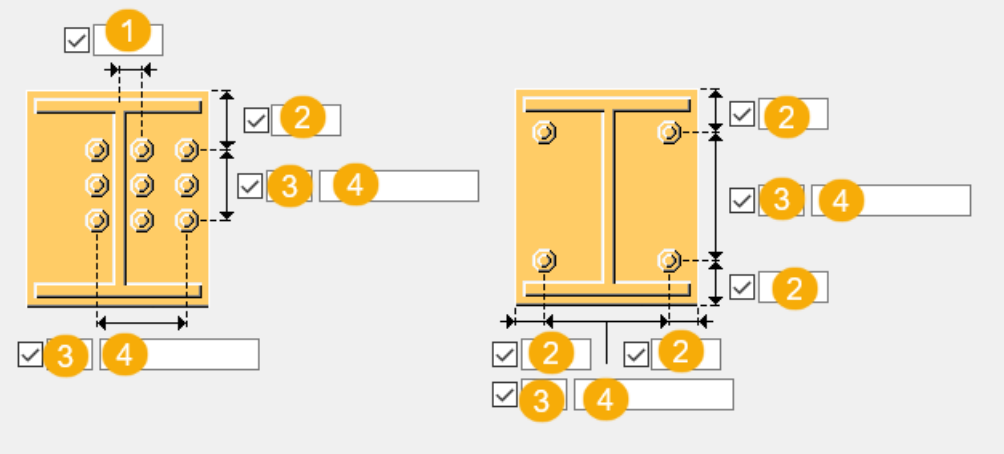
Depth measure

Option	Description
	Depth is calculated from the outer surfaces of the top and bottom flanges.
	Depth is the depth of the web.

Beam 2_2 tab

Use the **Beam 2_2** tab to define the end plate bolt group dimensions and bolt properties at the hip between the second beams.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

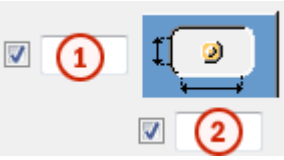
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Welds

Click the link below to find out more:

PEB Frame (S92)

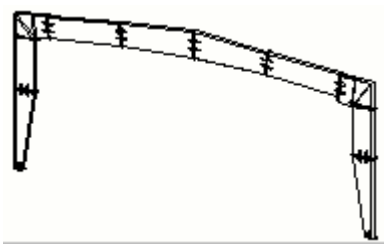
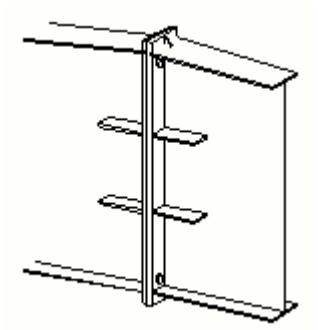
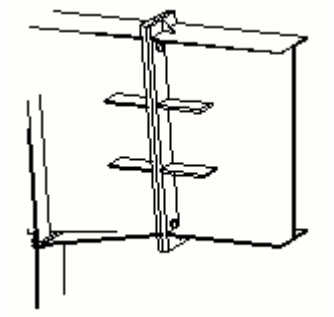
PEB Frame (S92) creates a pre-engineered building frame to the selected point. This component uses existing components to create built-up columns

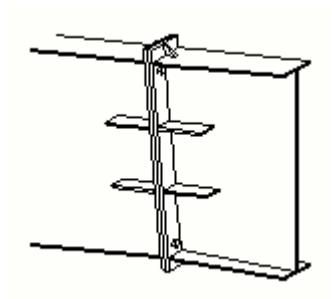
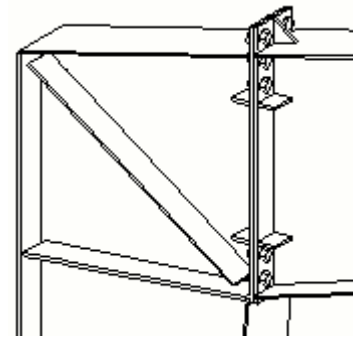
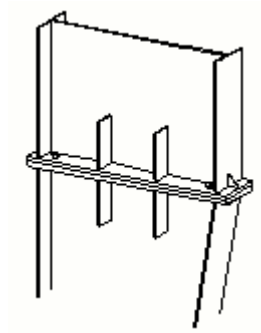
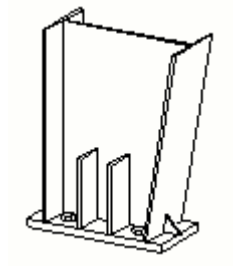
and rafters, base plates, knee connection, splice connections, and apex connection.

Objects created

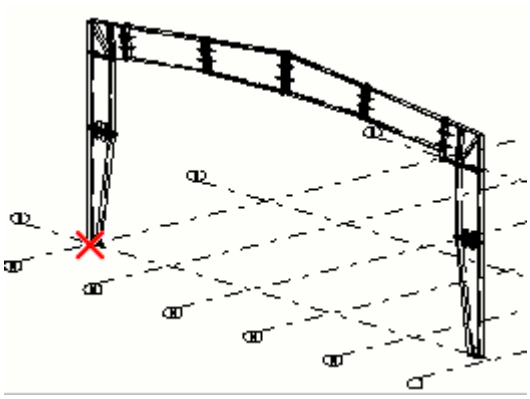
- Built-up rafter (1 to 4 segments)
- Apex connection
- Splice connections
- Knee connections (2)
- Built-up columns (1 to 4 segments)
- Base plate details (2)

Use for

Situation	Description
	Frame consisting of spliced columns and spliced rafters.
	Apex connection between two rafters at the apex of the portal frame.
	The first splice between the first and the second segment of the rafter.

Situation	Description
	The second splice between the second and the third segment of the rafter.
	Knee connection between the second segment of the column and the first segment of the rafter.
	Column splice between the first and the second segment of the column.
	Column base plate detail.

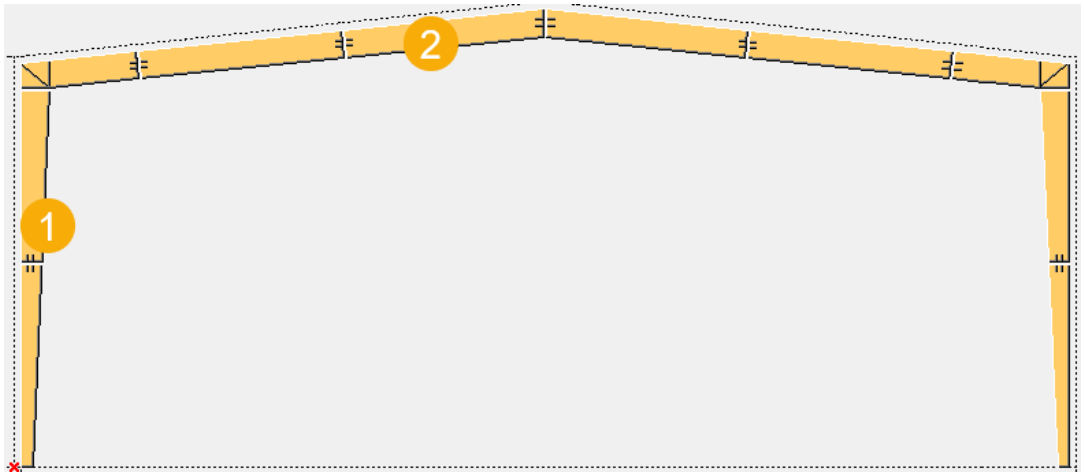
Selection order



1. Pick the point in which to insert the frame.

   The frame is created automatically when you pick the point.

Part identification key

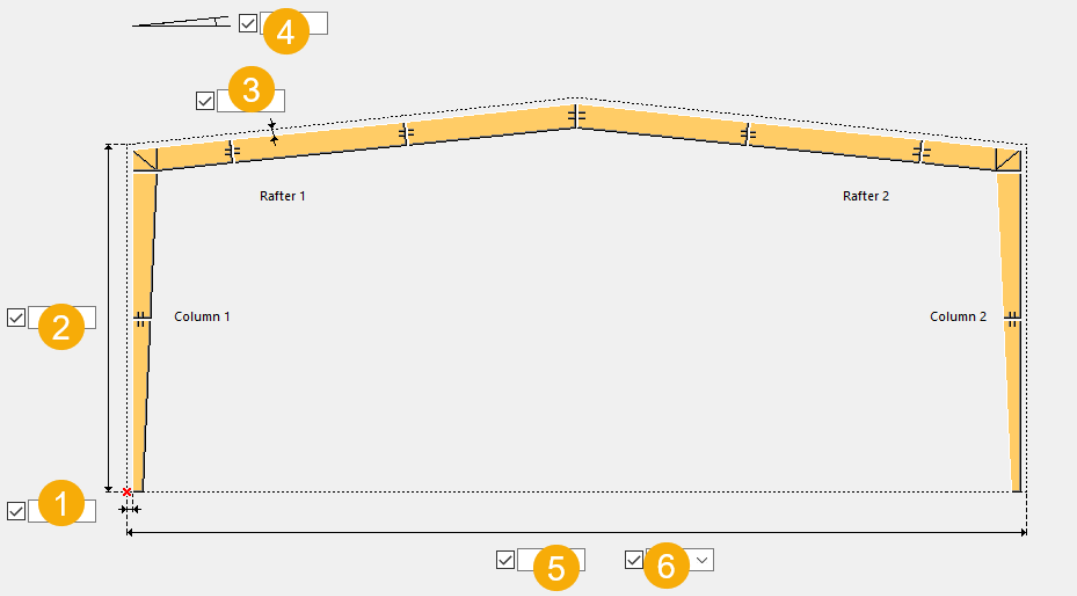


	Description
1	Column
2	Rafter

Picture tab

Use the **Picture** tab to define the frame dimensions.

Dimensions



	Description
1	Column horizontal distance from the input point
2	Column height
3	Rafter offset distance from the input frame
4	Rafter angle
5	Frame width
6	Orientation of the frame

Parameters tab

Use the **Parameters** tab to define part properties. Select whether the components are exploded or not.

Parts

Option	Description
Column 1, **Column 2**	Select a component from the **Applications & components** catalog. Define the segments and lengths.
Rafter 1, **Rafter 3**	Select a component from the **Applications & components** catalog. Define the segments and lengths.
Column 1 base plate, **Column 2 base plate**	Select a component from the **Applications & components** catalog.

Option	Description
Column 1 knee joint, **Column 2 knee joint**	Select a component from the **Applications & components** catalog.
Apex splice joint	Select a component from the **Applications & components** catalog.

Connections and details

Use the following components to automatically create connections and details for built-up beams and columns:

- Pipe column and beam panel zone (21) (page 2811)
- Batten plates (S85) (page 2824)
- Tapered column (136) (page 2831)
- Tapered beam to tapered column (197) (page 2840)
- Tapered column to tapered beam (199) (page 2845)
- Tapered beam to beam (200) (page 2851)
- Tapered column base plate (1068) (page 2859)
- PEB Knee joint (S93) (page 2877)

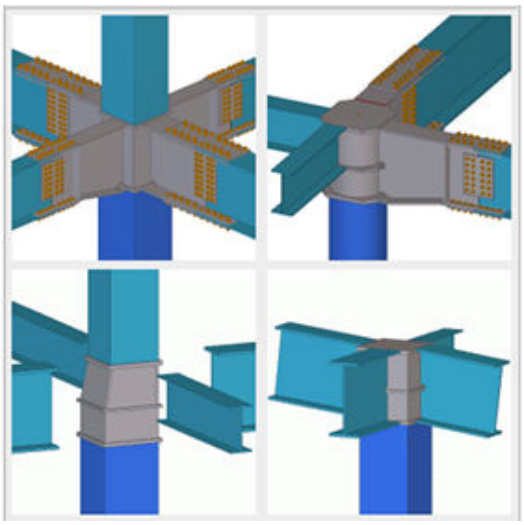
Pipe column and beam panel zone (21)

Pipe column and beam panel zone (21) connects a beam to a column with a diaphragm that goes through the column at the level of the beam flanges. You can use **Pipe column and beam panel zone (21)** both for square and round steel columns. H profile beams are used as secondary parts. The secondary part beams can be either horizontal or skewed.

Objects created

- Diaphragm (penetrating, inner)
- Brackets
- Cantilevers
- Welds

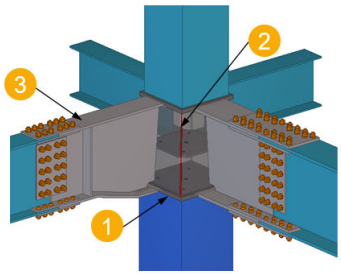
Use for

Situation	Description
	Beams connected to a column with brackets and cantilevers.

Selection order

1. Select the main part (lower column).
2. Select the secondary parts (a maximum of 4 beams).
3. Select the upper column as the fifth secondary part, if needed.
4. Click the middle mouse button to create the connection.

Part identification key

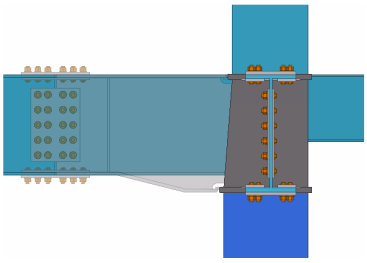
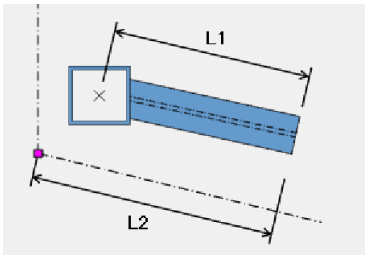


	Description
1	Diaphragm
2	Short columm Short columns are created between penetrating diaphragms.
3	**Tapered column-beam stub (150)** connection

Design dimensions tab

Use the **Design dimensions** tab to define the bracket type and length, and haunch height.

Bracket and hauch

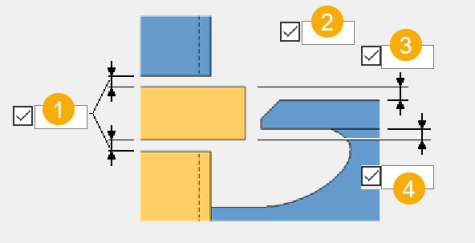
Option	Description	Default
Bracket type	Select the bracket type. • If you set the bracket type to **Bracket**, the **Tapered column-beam stub (150)** component is used for connecting the beam to the column. • If you set the bracket type to **Cantilever**, the beam is connected directly to the column. • If you set the bracket type to **Hypothetical**, the beam connection is created as diaphragm plates and short columns according to the beam size. 	**Bracket**
Bracket length	When you have set the bracket type to **Bracket**, select whether the bracket length is based on **Reference point (L2)** (this is the default) or **Center point (L1)**. The length is based on the lower column. 	1000 mm

Option	Description	Default
Haunch	Select whether the haunch height is automatically determined. You can use this option when the bracket type is **Bracket**.	**None**
Haunch height	Define the haunch height when the bracket type is not cantilever and you have set **Haunch** to **Specified**.	Straight bracket of the same size as the beam

Fabrication dimensions tab

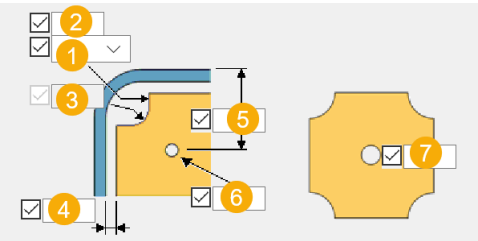
Use the **Fabrication dimensions** tab to define the diaphragm dimensions.

Dimensions



	Description	Default
1	Gap between the column and the diaphragm	0 mm
2	Maximum diaphragm thickness	40 mm
3	Outer additional diaphragm thickness	0 mm
5	Inner additional diaphragm thickness	0mm

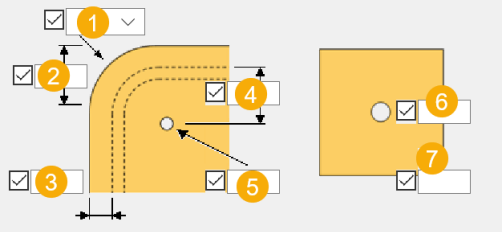
Inner diaphragm dimensions



	Description	Default
1	Chamfer shape of the inner diaphragm corner	Inner diaphragm: Concave circle Penetrating diaphragm: No chamfer

	Description	Default
2	Corner chamfer dimension	
3	Corner chamfer dimension for the extra arc	
4	Gap between the inner diaphragm and the inner surface of the column	0 mm
5	Spacing of the four corner holes of the inner diaphragm This option is used when the corner hole diameter defined in **6** is > 0 mm.	100 mm
6	Diameter of the corner hole that goes through the diaphragm There are four corner holes. You can define the diameter when the column is a square steel pipe.	0 mm
7	Diameter of the center hole that goes through the diaphragm	30 mm

Penetrating diaphragm dimensions

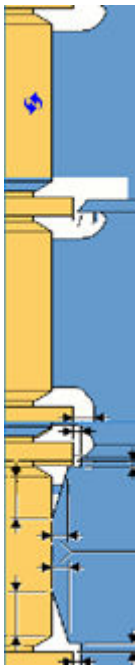


	Description	Description
1	Corner chamfer shape of the penetrating diaphragm	Inner diaphragm: Concave circle Penetrating diaphragm: No chamfer
2	Corner chamfer dimension You can define the dimension when the chamfer shape is set to something else than **No**.	0 mm
3	Diaphragm protrusion dimensions	25 mm
4	Spacing of the four corner holes of the through diaphragm This option is used when the corner hole diameter defined in **5** is > 0 mm.	100 mm

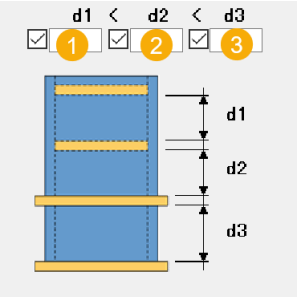
	Description	Description
5	Diameter of the corner hole that goes through the diaphragm There are four corner holes. You can define the diameter when the column is a square steel pipe.	
6	Diameter of the center hole that goes through the diaphragm	30 mm
7	Top plate is a penetrating diaphragm, called a top plate if there is no upper column.	

Weld access holes

You can define the weld access holes for the diaphragm plates.



Clearance dimensions



	Description	Default
1	Clearance between the penetrating diaphragms	150 mm

	Description	Default
2	Clearance between the inner diaphragms	100 mm
3	Clearance between the penetrating diaphragm and inner diaphragm	100 mm

Panel parts tab
Use the **Panel parts** tab to define the part properties.

Parts

Option	Description
Tapered plates	Thickness of the four tapered plates that build the short column. You can define the thickness when the taper type and straight short column type is **Buildup**.
Straight plates	Thickness of the four straight plates that build the short column. You can define the thickness when the taper type and straight short column type is **Buildup**.
Tapered profile	Select the profile from the profile catalog. Short column profile name can be used when the taper type and straight short column type is **Rolled**.
Straight profile	Select the profile from the profile catalog. Short column profile name can be used when the taper type and straight short column type is **Rolled**.

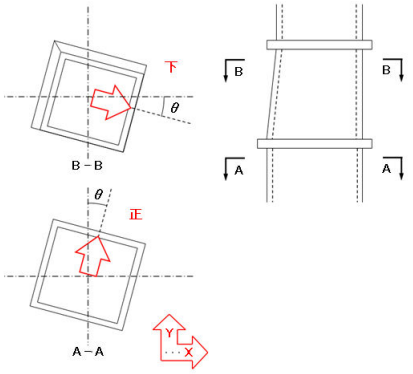
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part**

Option	Description	Default
		material box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Rotation of rolled steel

Rolled steel is placed at the center axis of the main part. If you use a rolled tapered profile, the taper may be eccentric. You then need to change the rotation by using the **Taper profile orientation** option.

Option	Description	Default
Panel plate size up	Increased panel thickness	0 increase
Taper panel	Select whether to taper the panel. You can define an additional rotation angle to the main part rotation angle.	**Auto**
Taper type	Select the tapering type.	**Buildup**
Taper profile orientation	Select the rotation direction of the tapered profile. The rotation follows the global coordinate system.	**Front**
Straight short column type	Select the short column type.	**Rolled**



Diaphragm thickness increase

Option	Regular plate size up	Plate size up	Size up due to column
Penetrating diaphragm **Inner diaphragm**	The default values are: • Penetrating diaphragm: 2 • Inner diaphragm: 1	To increase the size of a plate, set the size increase. You can define two size up values for each diaphragm plate, for a thin plate and a thick plate. For example, • Size up value for thin plate = 2 • Size up value for thick plate = 1 • The default border value = 40 mm If the required plate thickness is 34 mm, 2 size ups are applied and the result will be 45 mm. If the required plate thickness is 44 mm, 1 size up is applied and the result will be 50 mm.	Increased plate size The default values are: • Penetrating diaphragm: 1 • Inner diaphragm: 0

The table below shows an example of the calculation result.

Normal size up = 2	Specified plate thickness	Plate size up = 1
t ≦ 6 → 12 6 < t ≦ 9 → 16	< 40 ≦	t = 40 → 45 40 < t ≦ 45 → 48

Normal size up = 2	Specified plate thickness	Plate size up = 1
9 < t ≦ 12 → 19 12 < t ≦ 16 → 22 16 < t ≦ 19 → 25 … 32 < t ≦ 36 → 45		45 < t ≦ 48 → 50 48 < t ≦ 50 → 55 50 < t ≦ 55 → 60 …

Welds

Option	Description
Weld prep	Select whether to create welding cuts. The default is **No**.
Weld prep. preset	Select whether to use preset welding settings.

Custom component to column

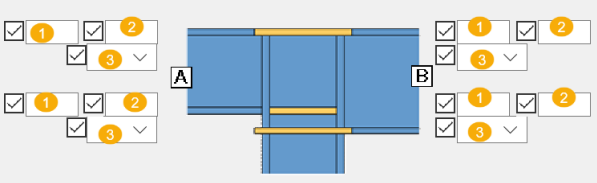
You can use custom components for welds, welding cuts, and weld backing bars.

Option	Description
Component	Custom component name You can use seam or connection type custom components.
Parameters	Property file name to be used for the custom component. You can define the name if you have specified a custom component.

Diaphragm tab

Use the **Diaphragm** tab to define the diaphragm material, thickness, offset, and type.

Diaphragm

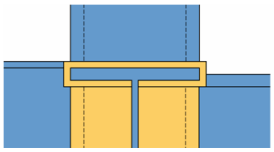


	Description
1	Diaphragm plate thickness
2	Offset from the automatically determined diaphragm position
3	Diaphragm type

Diaphragm thickness

Apply the highest value of the following options:

- Thickness is determined based on the thickest plate among the beam flanges that are at the same height.
- Thickness is determined by the thickest beam flange value + the additional thicknesses defined on the **Fabrication dimensions** tab.
- Thickness is calculated based on the required diaphragm thickness, which takes into account each beam flange thickness and height.

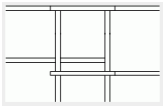
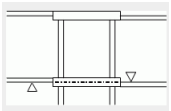


Diaphragm offset

The diaphragm offset is based on the automatically determined diaphragm height. The offset value is the distance from the reference height. Use the offset when fine-tuning the height of the automatically determined diaphragm. Enter a positive value for the upward offset and a negative value for the downward offset.

Diaphragm type

The diaphragm height is automatically determined by the height at which the beam flanges meet.

Example	Description
	The example image shows the most commonly used diaphragm arrangement. The outermost upper and lower diaphragms are penetrating diaphragms. The other diaphragm is an inner diaphragm.
	Even if two or more flanges are on slightly different levels, one diaphragm is created if the clearance is smaller than the minimum diaphragm clearance set on the **Fabrication dimensions** tab.

Cantilevers tab

Use the **Cantilevers** tab to define the custom components that are used.

Custom components

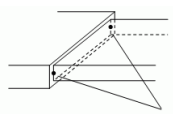
You can define that a custom component is used for the upper flange, lower flange, and the web.

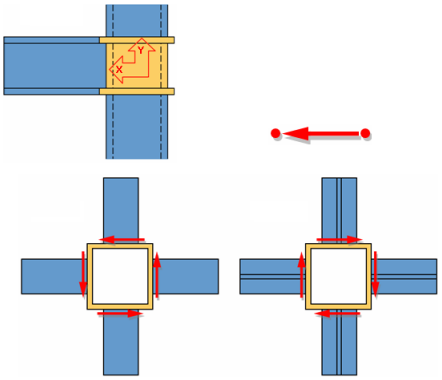
Component	Custom component name You can use seam or connection type custom components.
Parameters	Property file name to be used for the custom component. You can define the name if you have specified a custom component.

Custom components (upper flange, lower flange)

You can use custom components for welds, welding cuts, and weld backing bars.

The images below show examples of custom components applied to beam flanges.



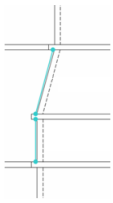


Custom components (web)

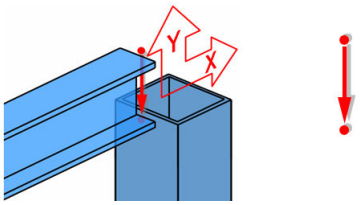
You can use custom components for polygon welds and web cuts.

The beam is the main part and the short column is the secondary part. If the web welds are separated by a penetrating diaphragm, apply a custom component to each web edge.

The example image below shows a situation where the penetrating diaphragm is interrupted and the side of the beam web is divided into two.



The example image below shows the coordinate system, and the first and second points used to define a seam type custom component.



Bracket tab

Use the **Bracket** tab to define the bracket properties.

Properties

Option	Description
Parameters	Property file name to be used for **Tapered column-beam stub (150)**. Define this when you have selected **Bracket** as the **Bracket type** on the **Design dimensions** tab.
Profile name	Select the profile from the profile catalog. The default value is the beam profile when the beam is rolled.
Size up	Plate size increase If you have selected a profile, the profile size value is used.
Profile/Flg	Select the profile/flange plate material.
Web	Select the web plate material. This option is not used when the bracket type is **Rolled**.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

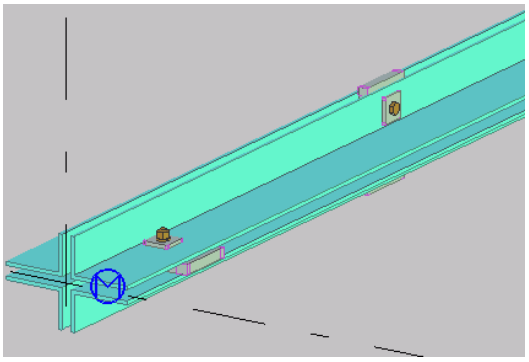
Batten plates (S85)

Batten plates (S85) connects twin profiles with either bolted or welded connections. Backing plates are also created.

Objects created

- Batten plates
- Backing plates
- Bolts
- Welds

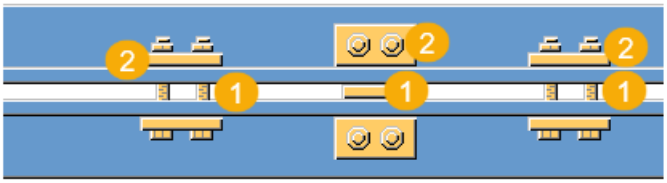
Use for

Situation	Description
	Twin profile connected with horizontal and vertical batten plates and backing plates.

Selection order

1. On the ribbon, click **Beam** --> **Twin profile**.
2. Pick a position on the twin profile.
3. Select the first object in the profile.
4. Select the second object in the profile.
5. Click the middle mouse button to create the connection.

Part identification key

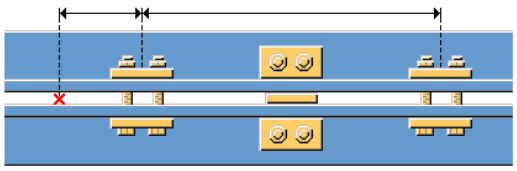
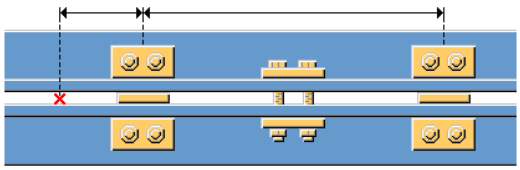
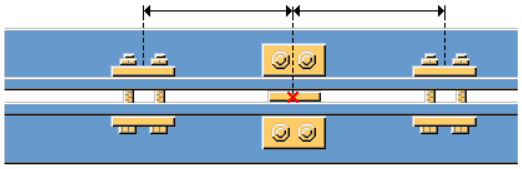
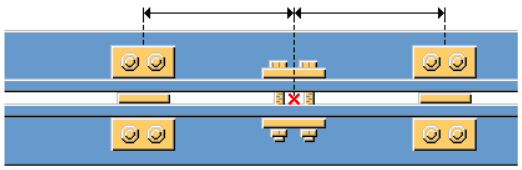


	Description
1	Batten plate
2	Backing plate

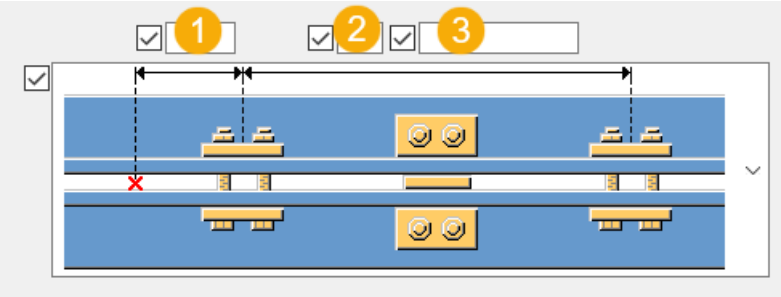
Picture tab

Use the **Picture** tab to define the distance of the batten plates from the input point, and the spacing and orientation of the plates.

Batten plate distribution

Option	Description
	The first batten plate is vertically oriented. Define the distance of the batten plates from the start point.
	The first batten plate is horizontally oriented. Define the distance of the batten plates from the start point.
	The batten plate in the middle is horizontally oriented. Define the distance of the batten plates from the middle point.
	The batten plate in the middle is vertically oriented. Define the distance of the batten plates from the middle point.

Batten plate location



	Description
1	Define the distance from the start point.
2	Define the number of batten plate connections.
3	Define the batten plate spacing.

Parts tab
Use the **Parts** tab to define the plate properties.

Parts

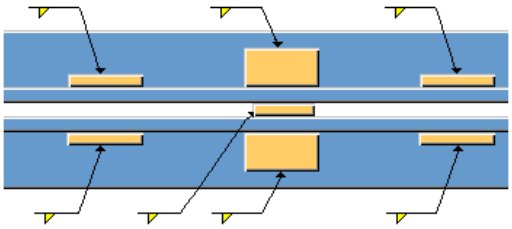
Option	Description	Default
Batten plate leg1	Thickness of the batten plate.	Clearance between the angles.
Batten plate leg2	Thickness of the batten plate.	Clearance between the angles.
Back plates	Thickness of the backing plates.	3/8"

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

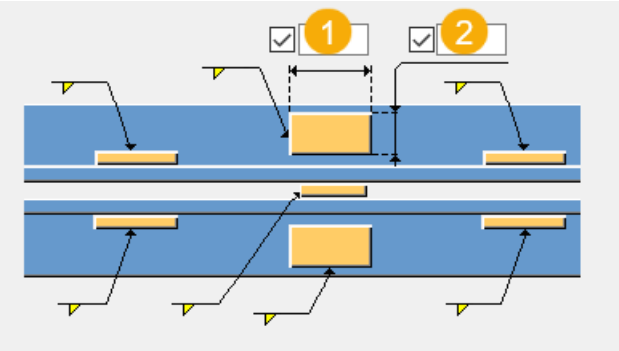
Parameters tab
Use the **Parameters** tab to select between bolted or welded plates. You can also define the size of the backing plates for the welded connections.

Plate type

Option	Description
	Plates are bolted.

Option	Description
	Plates are welded. Define the plate dimensions.

Welded plate dimensions

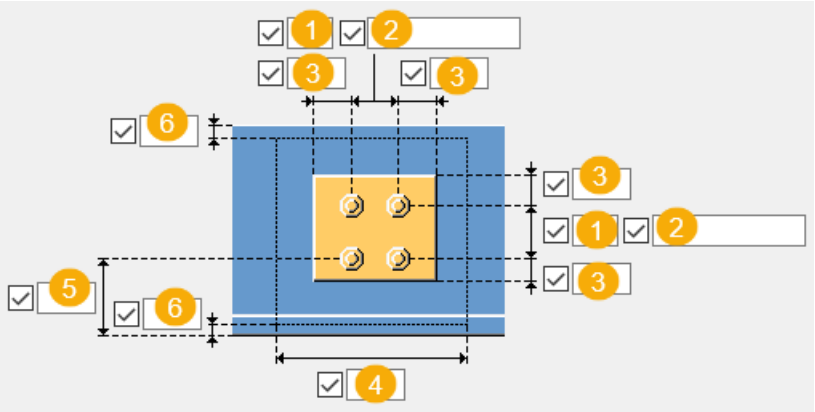


	Description	Default
1	Backing plate width.	3”15/16
2	Backing plate length.	3”15/16

Bolts tab

Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions



	Description
1	Number of bolts.

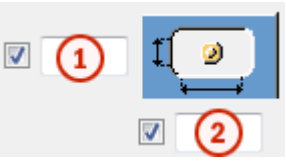
	Description
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Batten plate width.
5	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
6	Batten plate edge distance from the part edge.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

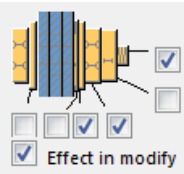


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Welds

Click the link below to find out more:

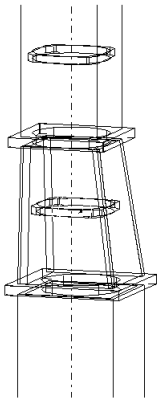
Tapered column (136)

Tapered column (136) creates a transition plate between different size columns. The transition plates can either be tapered or straight.

Objects created

- Diaphragm plates (penetrating and inner)
- Transition plates

Use for

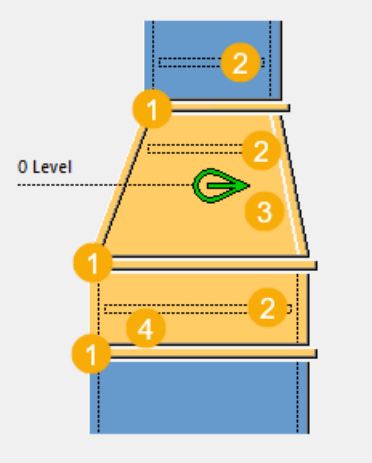
Situation	Description
	Penetrating and inner diaphragm plates, and a tapered section are created.

Selection order

1. Select the main part (lower part).
2. Select the secondary part (upper part).

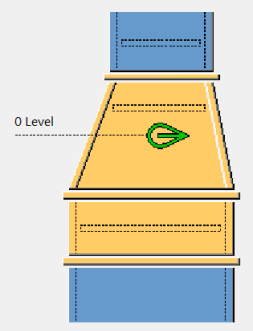
   The connection is created automatically when the secondary part is selected.

Part identification key



	Description
1	Penetrating diaphragm
2	Inner diaphragm
3	Transition plate (tapered)
4	Transition plate (straight)

Picture tab
The **Picture** tab illustrates the elevation to which all of the parts created by the connection are relative to.



Parameters tab
Use the **Parameters** tab to define the type of transition columns used in the connection. The short sections of columns can be created of rolled tubes or built up from plates.

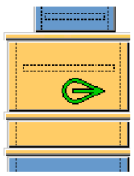
Parts

Option	Description
Penetrating diaphragm, **Inner diaphragm**	Define the thickness on the **Diaphragm** tab.
Tapered section	Thickness of the section.
Buildup section	Thickness of the section.
Upper rolled col	Select the profile from the profile catalog.
Lower rolled col	Select the profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in

Option	Description	Default
		File menu --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Tapered section position

	Default Tapered section is in the upper location. AutoDefaults can change this option.
	Tapered section is not created.
	Tapered section is in the upper location.
	Tapered section is in the lower location.

Straight short column type

	Default Short straight column is created using the defined rolled profile. AutoDefaults can change this option.

	Short straight column is created using the defined rolled profile.
	Short straight column is created using the defined built-up plate.

Solid weld cut

Solid weld cut creates beveled cuts for groove welds in the built-up and tapered column plate edges.

	Default Beveled edge cuts are not created. AutoDefaults can change this option.
	Beveled edge cuts are created.
	Beveled edge cuts are not created.

Diaphragm tab

Use the **Diaphragm** tab to set the number, position and type of the diaphragm plates created in the connection.

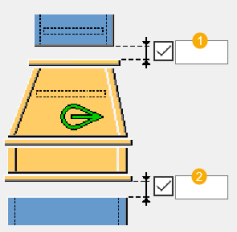
Option	Description	Default
t	Diaphragm plate thickness.	No default Enter a plate thickness to create the diaphragm plate.

Option	Description	Default
Level	Relative distance from the elevation level shown on the **Picture** tab.	Enter a plus or minus dimension to locate the desired diaphragm.
Depth	Position of the diaphragm in relation to the value entered in the **Level** box.	
Offset	Diaphragm plate offset from the value entered in the **Level** box.	0"
Diaphragm	Diaphragm type.	
Bevel cut	Position of the bevel cut on the diaphragm plate.	

Penetrating diaphragm

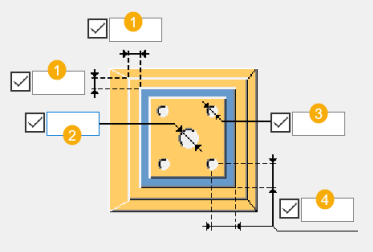
Use the **Penetrating diaphragm** tab to control the size and shape of the diaphragm plate. You can also add holes to the plate and set the root distance from the diaphragm to the column.

Welding gap



	Description	Default
1	Welding gap to the upper column. Distance from the column to the diaphragm plate.	0"
2	Welding gap to the tower column. Distance from the column to the diaphragm plate.	0"

Dimensions

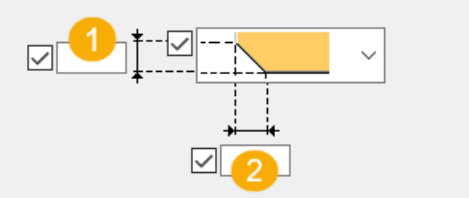


	Description	Default
1	Diaphragm plate edge distance outside of the face of column.	1"
2	Diameter of the center hole in the diaphragm plate.	13/16"
3	Diameter of corner holes created in each corner of the diaphragm plate. To create corner holes you must enter a value.	
4	Position of the corner holes in relation to the face of column.	1"3/16

Chamfer type

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Chamfer dimensions

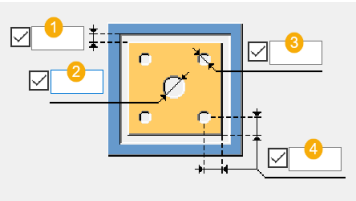


	Description
1	Vertical chamfer dimension.
2	Horizontal chamfer dimension.

Inner diaphragm

Use the **Inner diaphragm** tab to control the size of the diaphragm plate. You can also add holes to the plate and chamfer the corners.

Dimensions



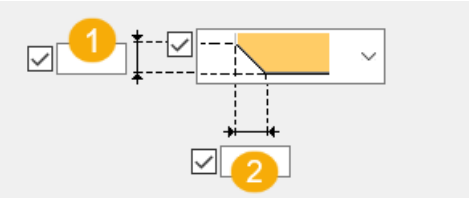
		Defua
	Weld gap Distance from the inside face of the column to the diaphragm plate.	
	Diameter of the center hole in the diaphragm plate.	13/16"
	Diameter of the corner holes created in each corner of the diaphragm plate. To create corner holes you must enter a value.	
	Position of the corner holes in relation to the edge of the inner diaphragm plate.	1"3/16

Chamfer type

Option	Description
	Default No chamfer AutoDefaults can change this option.

Option	Description
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Chamfer dimensions



	Description
1	Vertical chamfer dimension.
2	Horizontal chamfer dimension.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

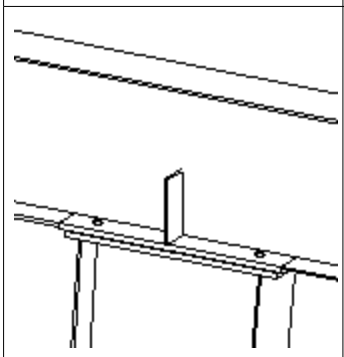
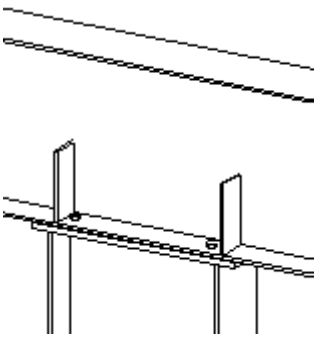
Tapered beam to tapered column (197)

Tapered beam to tapered column (197) creates a connection between a tapered beam and a tapered column.

Objects created

- End plates (column top, beam flange)
- Stiffeners (2 or 4)
- Bolts
- Welds

Use for

Situation	Description
	Connection is created using two stiffeners, and flange and column plates. Plates are connected with four bolts, but you can use any number of bolts.
	Connection is created using four stiffeners and a column end plate. A separate plate for the beam flange is not created.

Before you start

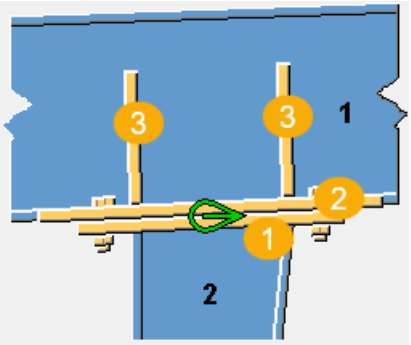
- Create a tapered beam.
- Create a tapered column.

Selection order

1. Select the flange of the main part (beam).
2. Select the web of the secondary part (column).

   The connection is created automatically when the secondary part is selected.

Part identification key

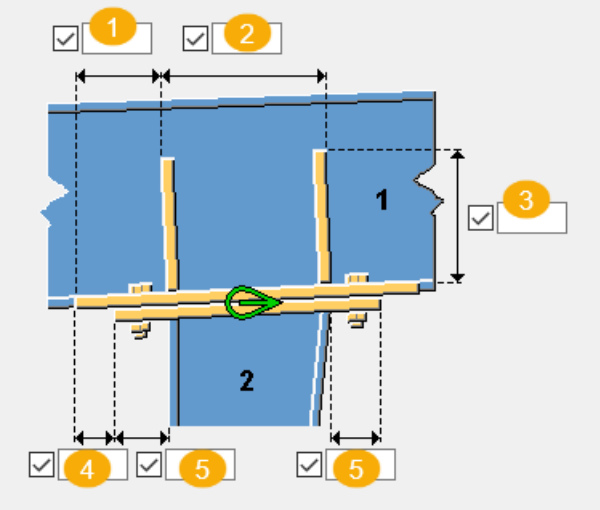


	Description
1	Column end plate
2	Beam flange end plate
3	Stiffener

Picture tab

Use **Picture** tab to define the dimensions of end plates and stiffeners, and the location of stiffeners.

Dimensions



	Description
1	Beam end plate edge distance to the stiffener edge.
2	Distance between the stiffeners.
3	Stiffener dimension from the beam web.
4	Column end plate edge distance to the beam end plate edge.
5	Column end plate edge distance to the column edge.

Parts tab
Use the **Parts** tab to define the properties of the column end plate, the beam end plate, and the stiffeners.

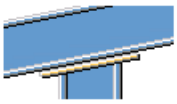
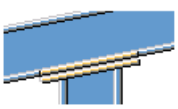
Parts

Option	Description
Beam plate	Thickness and width of the beam plate.
Column plate	Thickness and width of the column plate.
Stiffener	Thickness and width of the stiffener.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab
Use the **Parameters** tab to define the plate and stiffener positions and orientation.

Plates

Option	Description
	End plate is created at the column end.
	End plated is created at the column end and beam flange.

Stiffeners

Option	Description
	One stiffener is created.
	Two stiffeners are created.

Column flange fit

Option	Description
	Column flange is fitted to be level with the end plate.
	Column flange is perpendicular to the column axis.

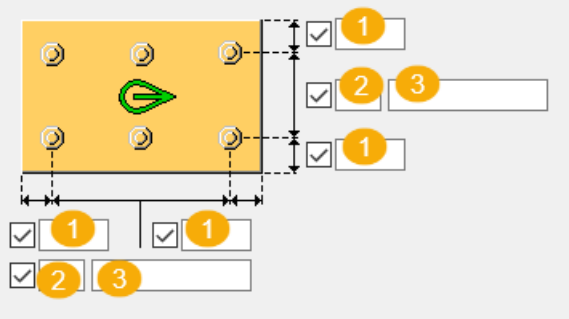
Stiffener orientation

Option	Description
	Stiffener is parallel to the main part.
	Stiffener is perpedicular to the beam.

Bolts tab

Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions

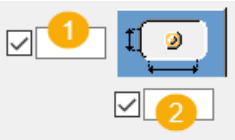


	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

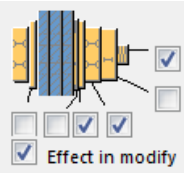


	Description
1	Vertical dimension of slotted hole.
2	Horizontal dimension of slotted hole.

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Tapered column to tapered beam (199)

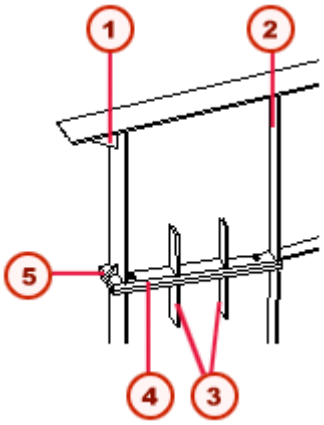
Tapered column to tapered beam (199) creates a connection between a tapered column and a tapered beam. All parts are optional.

Objects created

- End plates (column top, beam flange)
- Web stiffeners (8)
- Triangular beam flange stiffener

- Triangular flange stiffeners (2)
- Beam web stiffeners (2)
- Bolts
- Welds

Use for

Situation	Description
	Connection is created using all possible parts. Plates are connected using four bolts, but you can use any number of bolts. **1** Beam flange stiffener **2** Beam web stiffeners **3** Web stiffeners **4** End plates **5** Flange stiffeners

Before you start

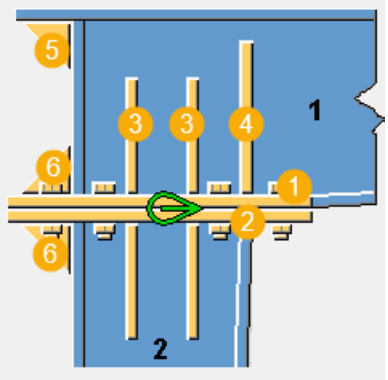
- Create a tapered beam.
- Create a tapered column.

Selection order

1. Select the web of the main part (beam).
2. Select the web of the secondary part (column).

   The connection is created automatically when the secondary part is selected.

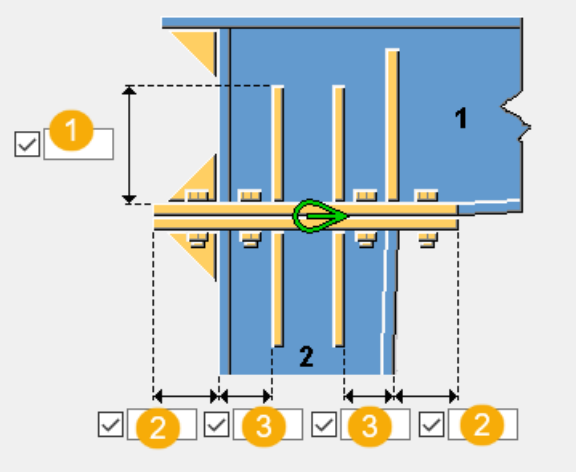
Part identification key



	Description
1	Beam end plate
2	Column end plate
3	Web stiffeners
4	Beam web stiffener
5	Beam flange stiffener
6	Flange stiffener

Picture tab
Use the **Picture** tab to define the dimensions of end plates and stiffeners, and the location of stiffeners.

Dimensions



	Description
1	Web stiffener dimension from the end plate.
2	End plate edge dimension from the secondary part edge.
3	Stiffener edge dimension from the secondary part edge.

Parts tab
Use the **Parts** tab to define the properties of the column plate, beam plate, and stiffeners. Set the part thickness to `0` to omit a part.

Parts

Option	Description
End plate	Thickness and width of the end plate.
Web stiffener	Thickness and width of the web stiffener.

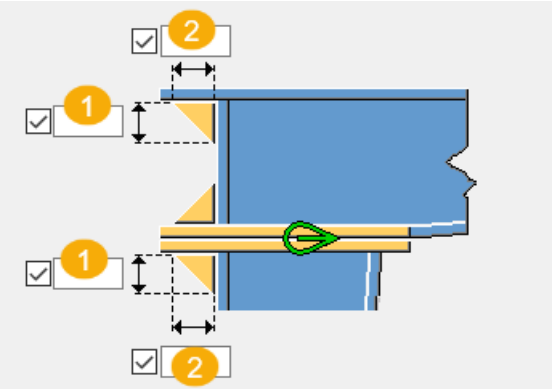
Option	Description
Flange stiffener	Thickness of the flange stiffener.
Beam flange stiff	Thickness of the beam flange stiffener.
Beam web stiff	Thickness and width of the beam web stiffener.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab

Use the **Parameters** tab to define whether the column flange and the corner are fitted, and the stiffener chamfer dimensions.

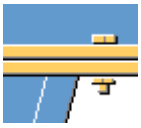
Stiffener chamfer dimensions



	Description
1	Vertical chamfer dimension.

	Description
2	Horizontal chamfer dimension.

Column flange fit

Option	Description
	Column flange fitted to be level with the end plate.
	Column flange is perpendicular to the column axis.

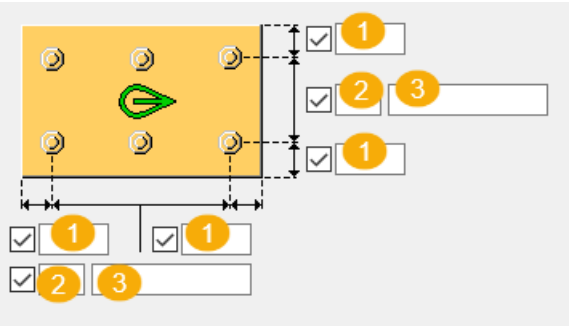
Corner fitting

Option	Description
	Corner is fitted.
	Corner is not fitted.

Bolts tab

Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

Bolts group dimensions



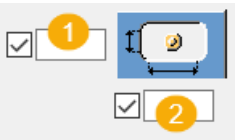
	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.

	Description
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

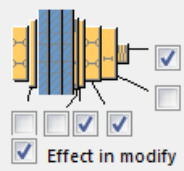


	Description
1	Vertical dimension of slotted hole.
2	Horizontal dimension of slotted hole.

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

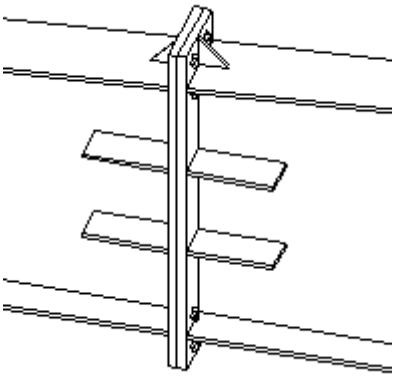
Tapered beam to beam (200)

Tapered beam to beam (200) creates a connection between tapered beams.

Objects created

- End plates (2)
- Flange stiffeners (4) (optional)
- Web stiffeners (8) (optional)
- Bolts
- Welds

Use for

Situation	Description
	All stiffeners and end plates are created. Plates are connected with eight bolts, but you can use any number of bolts.

Before you start

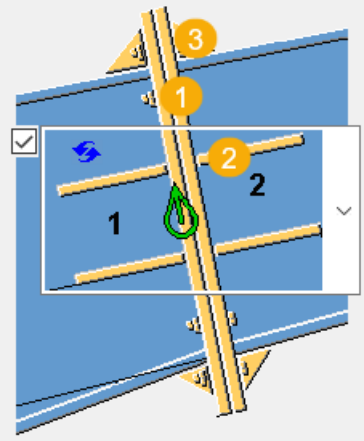
Create two tapered beams.

Selection order

1. Select the web of the main part (first beam).
2. Select the web of the secondary part (second beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

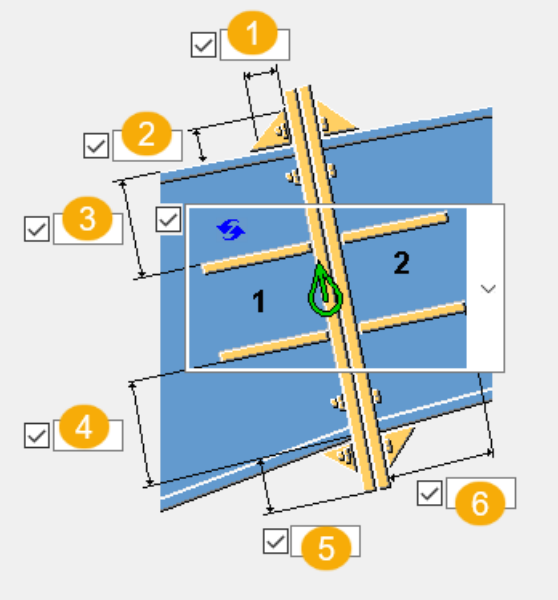


	Description
1	End plates
2	Web stiffeners
3	Flange stiffeners

Picture tab

Use the **Picture** tab to define the dimensions of the end plates and stiffeners, the location of the stiffeners, and whether the beam flange is fitted.

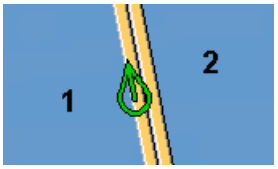
Dimensions



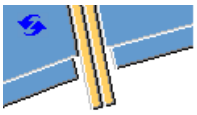
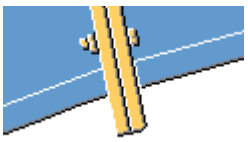
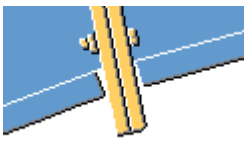
	Description
1	Horizontal dimension of the flange stiffener.
2	Vertical dimension of the flange stiffener.
3	Top web stiffener edge distance to the beam flange.
4	Bottom web stiffener edge distance to the beam flange.
5	End plate dimension from the beam edge.
6	Web stiffener dimension from the end plate.

Web stiffeners

Option	Description
	Default Top and bottom web stiffeners are created. AutoDefaults can change this option.
	Top and bottom web stiffeners are created.

Option	Description
	Top web stiffeners are created.
	Bottom web stiffeners are created.
	Web stiffeners are not created.

Beam flange

Option	Description
	Default Beam flanges are perpendicular to the beam axis. AutoDefaults can change this option.
	Beam flanges are fitted to be level with the end plat
	Beam flanges are perpendicular to the beam axis.

Parts tab

Use the **Parts** tab to define the properties of end plates, web stiffeners, and flange stiffeners. Set the part thickness to `0` to omit a part.

Parts

Option	Description
End plate	Thickness and width of the end plate.
Web stiffener	Thickness and width of the web stiffener.

Option	Description
Flange stiffener	Thickness of the flange stiffener.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Chamfer tab
Use the **Chamfer** tab to define the flange and web stiffener shape, type, and dimensions.

Flange stiffener

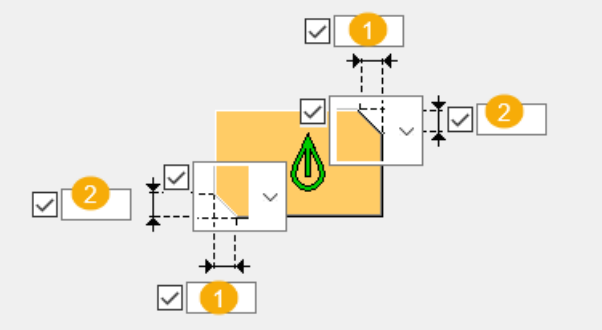
Option	Description
	Default Bevel, chamfered corner AutoDefaults can change this option.
	Bevel, chamfered corner
	Square, one chamfered corner
	Square, two chamfered corners

Chamfer type

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Chamfer dimensions

You can define the chamfer dimensions for both web and flange stiffeners.

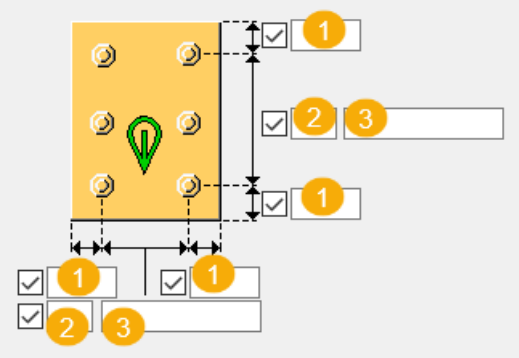


	Description
1	Horizontal chamfer dimension.
2	Vertical chamfer dimension.

Bolts tab

Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions



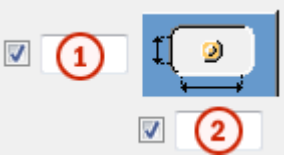
	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

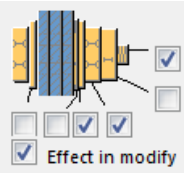


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

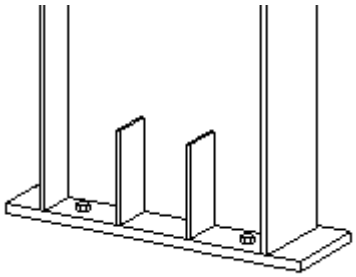
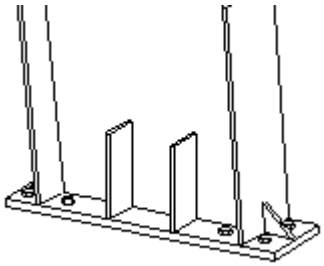
Tapered column base plate (1068)

Tapered column base plate (1068) creates a base plate detail to a tapered column.

Parts created

- End plate
- Web stiffeners (4)
- Flange stiffeners (2) (optional)
- Anchor rods
- Extra plates connecting the anchor rods
- Bolts
- Welds

Use for

Situation	Example
	The detail uses four web stiffeners. Flange stiffeners are not created. The end plate is connected using four bolts, but you can use any number.
	The detail uses four web stiffeners and two flange stiffeners.

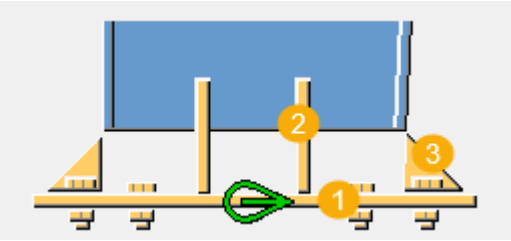
Before you start

Create a tapered column.

Selection order

1. Select the tapered column.
2. Pick a point to indicate the location of base plate.

Part identification key

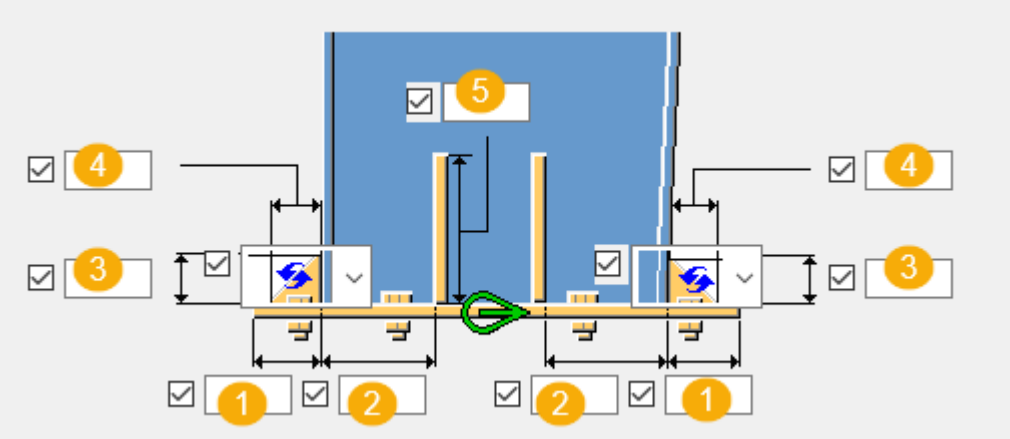


	Description
1	End plate
2	Web stiffener
3	Flange stiffener

Picture tab

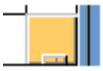
Use the **Picture** tab to define the dimensions of the base plate and stiffeners, the location of web stiffeners, and whether the column flange is fitted.

Dimensions

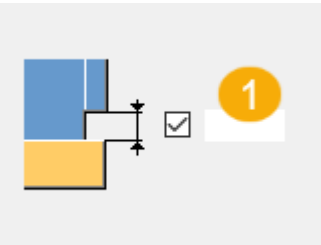


	Description
1	End plate edge distance to the column flange.
2	Web stiffener edge distance.
3	Flange stiffener height.
4	Flange stiffener width.
5	Web stiffener height.

Flange stiffener shape

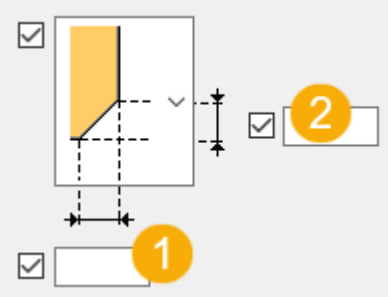
Option	Description
	Default Line shape AutoDefaults can change this option.
	Line shape
	Square shape
	No stiffener

Flange cut



	Description	Default
1	Define the flange cut from the base plate edge.	2 mm

Flange stiffener chamfer dimensions

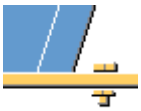


	Description
1	Horizontal chamfer dimension.
2	Vertical chamfer dimension.

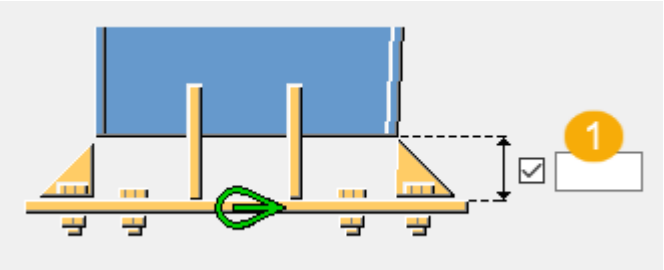
Flange stiffener chamfer type

Option	Description
	Default No chamfers AutoDefaults can change this option.
	No chamfers
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Column flange

Option	Description
	Column flange fitted to be level with the base plate.
	Column flange is perpendicular to column axis.

End plate offset



	Description
1	End plate offset from the main part bottom edge.

Parts tab

Use the **Parts** tab to define the properties of the end plate, web stiffeners, and flange stiffeners.

Parts

Option	Description
End plate	Thickness and width of the end plate.
Web stiffener	Thickness and width of the web stiffener.
Flange stiffener	Thickness of the flange stiffener.

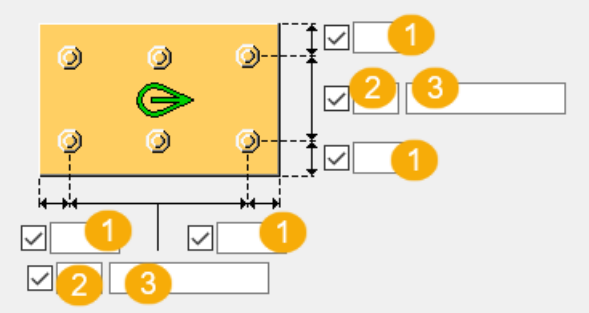
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Bolts tab

Use the **Bolts** tab to define the bolt properties, number of bolts, and spacing.

Bolt group dimensions



	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.

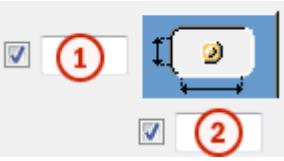
Option	Description	Default
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Slotted holes

You can define slotted, oversized, or tapped holes.



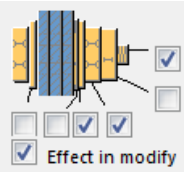
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options	

Option	Description	Default
	depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

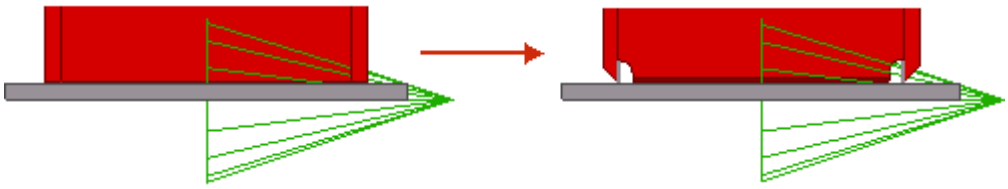
Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Parameters tab
Use the **Parameters** tab to define a component.

Using additional components

You can use additional system or custom components to modify the column end or the base plate. For example, you can create special backing plates, weld preparations, and weld access holes for the column end.

If you use additional system or custom components, you need to manage the column end or the base plate properties in the additional component in question. When using several components, there may be multiple welds and cuts.



Option	Description
Component	Define a system or a custom component by selecting it from the component catalog.
Attributes	Enter the name of the attribute file for the selected component.
Input	Define to which parts the selected component is applied. • **Default** is the same as **Base + Column**. • **Column** sets the column as the main part. Use this option for details. • **Column + Base** sets the column as the main part and the base plate as the secondary part. • **Base + Column** sets the base plate as the main part and the column as the secondary part. • **Base** sets the base plate as the main part.

Anchor rods tab

Use the **Anchor rods** tab to define the properties of the rod, nut, washer, cast plate, and grout.

Anchor rod dimensions

Option	Description
Rod profile	Anchor rod profile. You can add a comment about the part.
Nut profile	Nut profile.
Washer profile	Washer profile.
Plate washer	Plate washer thickness, width and height.
Cast plate	Cast plate thickness, width and height.
Grout	Grout thickness. Grouting helps you to model columns to the top of concrete parts and place the base plate correctly. It also makes

Option	Description
	it easier to dimension the detail in GA drawings. By default, no grouting is created. Select whether the grouting is created with or without slopes above or below the detail creation point. This also affects the shim plates.

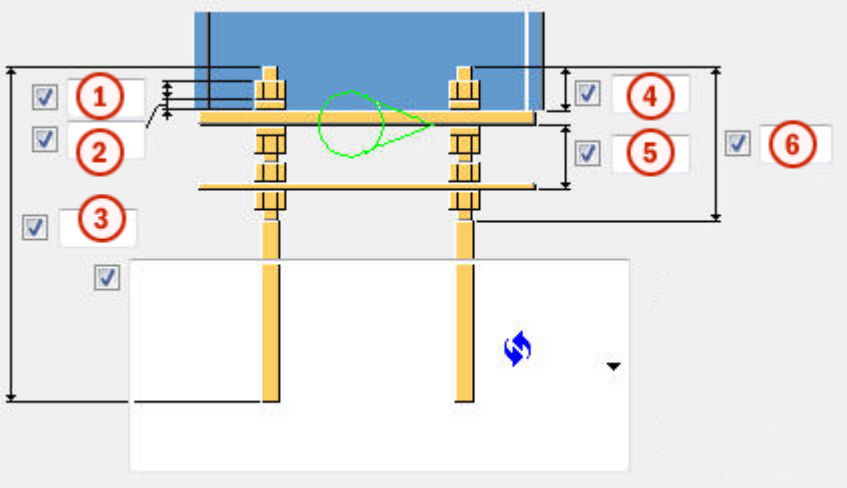
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

Base plate with

Select whether to create the base plate with bolts, anchor rods, or a custom component.

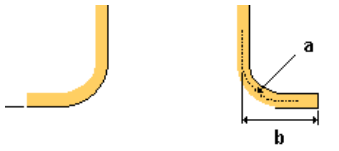
By default, the base plate is created with **Bolts**.

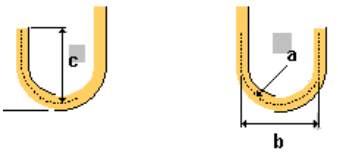
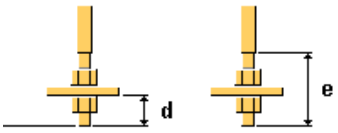
Anchor rod dimensions



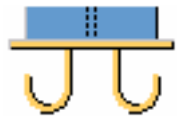
	Description
1	Size or the length of the nut.
2	Size or the thickness of the washer.
3	Length of the anchor rod.
4	Length of the anchor rod above the base plate.
5	Distance between the cast plate and the base plate.
6	Length of the upper thread.

Anchor rod types

Option	Description	
	Default Type 1 AutoDefaults can change this option.	
	Type 1	
	**a** Radius of the hook **b** Width of the hook	**a** = `2*anchor bar diameter` **b** = `1/5 of anchor bar length`

Option	Description	
	**a** Radius of the hook **b** Width of the hook **c** Height of the hook	**c** = `same as width of the hook`
	**d** Length of the anchor rod below the extra plate **e** Length of the lower thread	**d** = `2*nut size` **e** = `4*nut size plus thickness of extra plate`

Hook direction

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2
	Type 3
	Type 4

Bolting direction

You can define the bolting direction if you have created the base plate with bolts.

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1
	Bolting direction 2

Cast plate holes tolerance

Option	Description	Default
Cast plate holes tolerance	Tolerance of the cast plate holes.	`same as bolt tolerance`

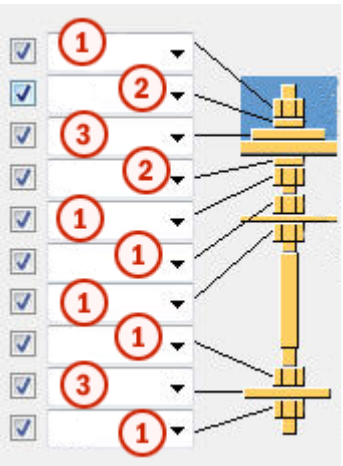
Washer hole tolerance

Option	Description
Create hole in washer	By default, a hole is not created in the washer. Tolerance of the washer hole.

Create assembly from all anchors

Define whether anchors are included in an anchor rod assembly. You can also include leveling plates into the assembly.

Create



1	Create the nut profile.
2	Create the washer profile.
3	Create the washer plate.

Anchor rod assembly

Define which parts of the anchor rod are included in the anchor rod assembly.

You can weld the washer plates above and below the base plate.

Extra plates tab

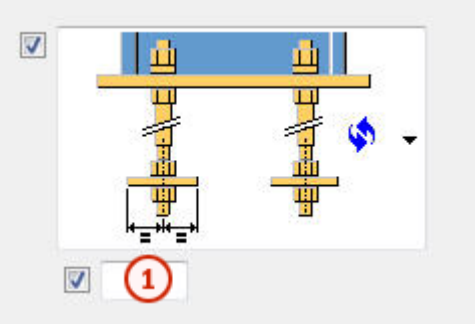
Use the **Extra plates** tab to to control the placement, rotation, and type of the profiles (extra profile 1) created at the bottom of each anchor bar and the profiles (extra profile 2) that connect rows of anchor bars.

Part dimensions

Option	Description
Extra profile 1	First extra profile by selecting it from the profile catalog.
Extra profile 2	Second extra profile by selecting it from the profile catalog.

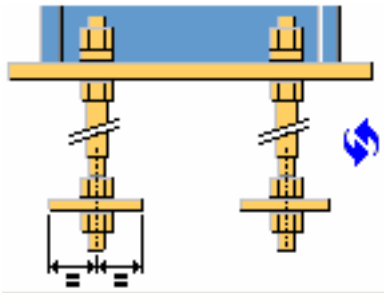
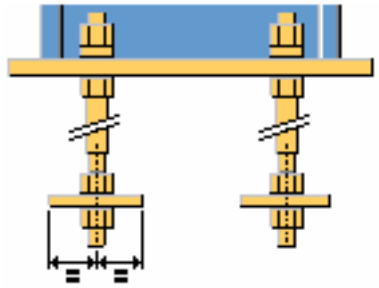
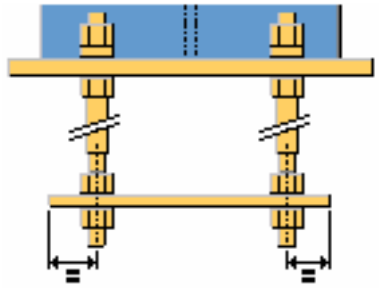
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Finish	Describes how the part surface has been treated.	

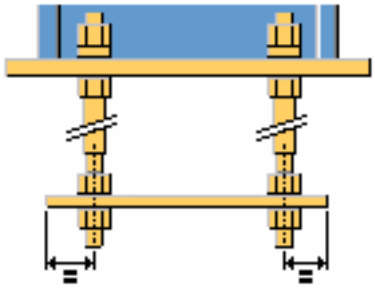
Edge distance of extra profile 1



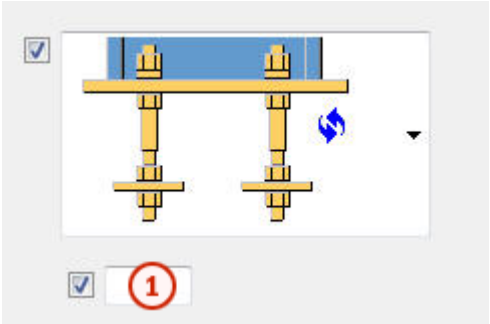
	Description
1	Edge distance of extra profile 1.

Type and direction of extra profile 1

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2

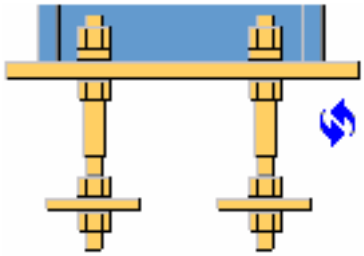
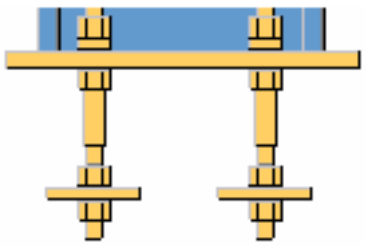
Option	Description
	Type 3

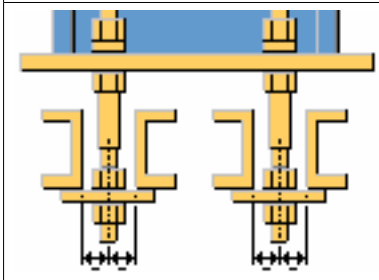
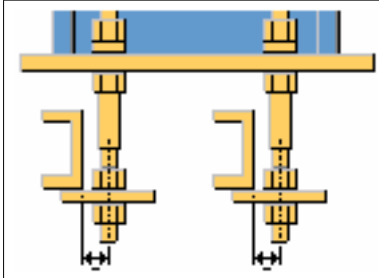
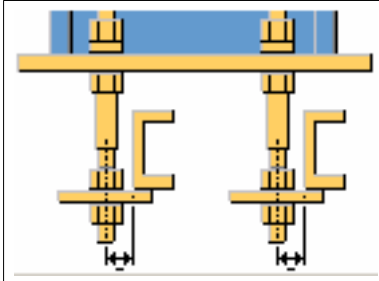
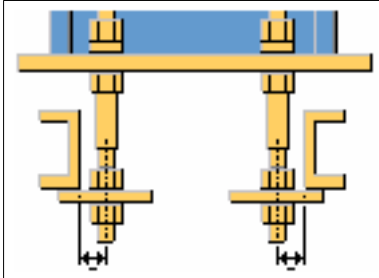
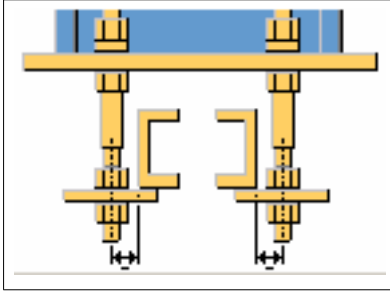
Edge distance of extra profile 2



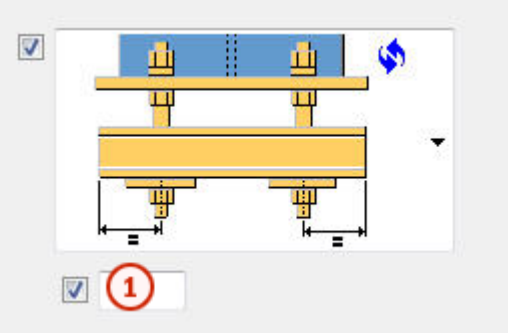
Option	Description	Default
1	Distance of extra profile 2 from the axis of the anchor bar.	`Half of the nut size or anchor bar diameter`

Extra profile 2 type

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1

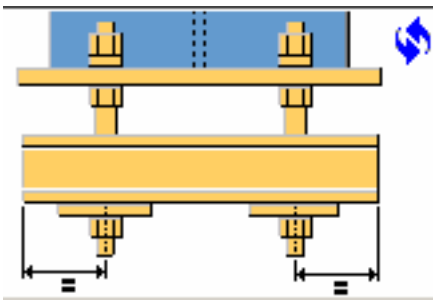
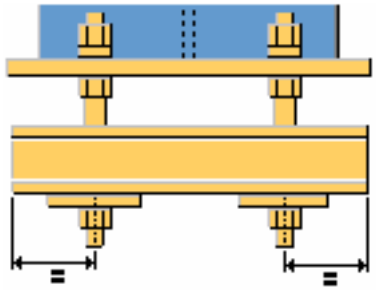
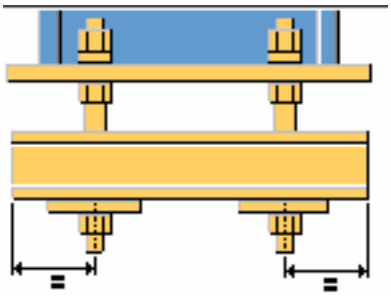
Option	Description
	Type 2
	Type 3
	Type 4
	Type 5
	Type 6

Length of extra profile 2



	Description
1	Length of extra profile 2 from the axis of the anchor bar.

Direction of extra profile 2

Option	Description
	Default Type 1 AutoDefaults can change this option.
	Type 1
	Type 2

Extra profile 1 properties

Option	Description	Default
Hole tolerance	Hole tolerance of extra profile 1.	`Same as bolt tolerance`
Circular profile height	Height of circular extra profile 1.	
Profile rotation	Profile rotation of extra profile 1.	`Front`

Extra profile 2 rotation

Option	Description	Default
Extra profile 2 rotation	Profile rotation of extra profile 2.	`Front`

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

PEB Knee joint (S93)

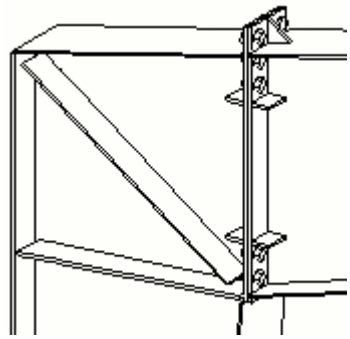
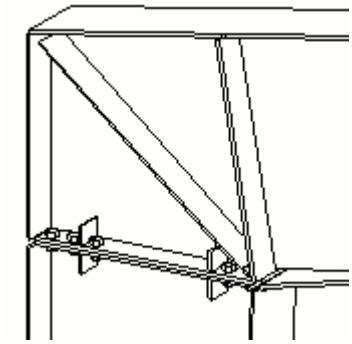
PEB Knee joint (S93) creates a knee connection between a rafter and a column. The end plate connection can be located horizontally, vertically, or perpendicular to the rafter.

Objects created

- End plates
- Web stiffener
- Flange stiffeners (2)
- Inner stiffeners (4)
- Outer stiffeners (4)
- Diagonal stiffeners or web plate

- Bolts
- Welds

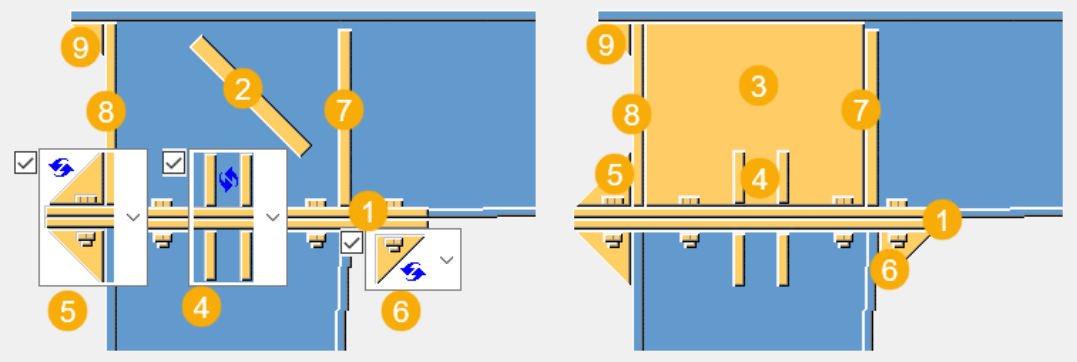
Use for

Situation	Description
	Vertical knee connection with triangular flange stiffeners, and inner and outer stiffeners. Web is strengthened with diagonal stiffeners.
	Horizontal knee connection with inner and outer stiffeners. Web is strengthened with diagonal stiffeners.

Selection order

1. Select the column.
2. Select the rafter.
3. Click the middle mouse button to create the connection.

Part identification key



	Description
1	End plate
2	Diagonal stiffener
3	Web plate
4	Web stiffener
5	External stiffener
6	Inner stiffener
7	Knee stiffener
8	Knee end plate
9	Corner lap stiffener

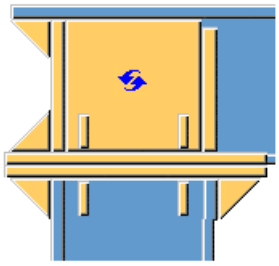
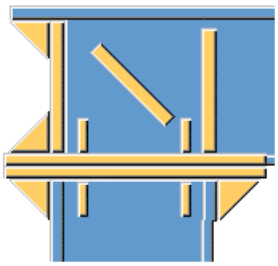
Picture tab
Use the **Picture** tab to define the dimensions and the connection type.

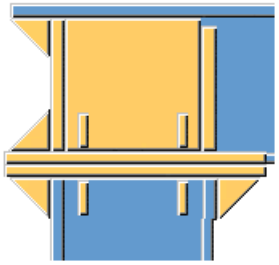
Connection type

Select how the connection is created:

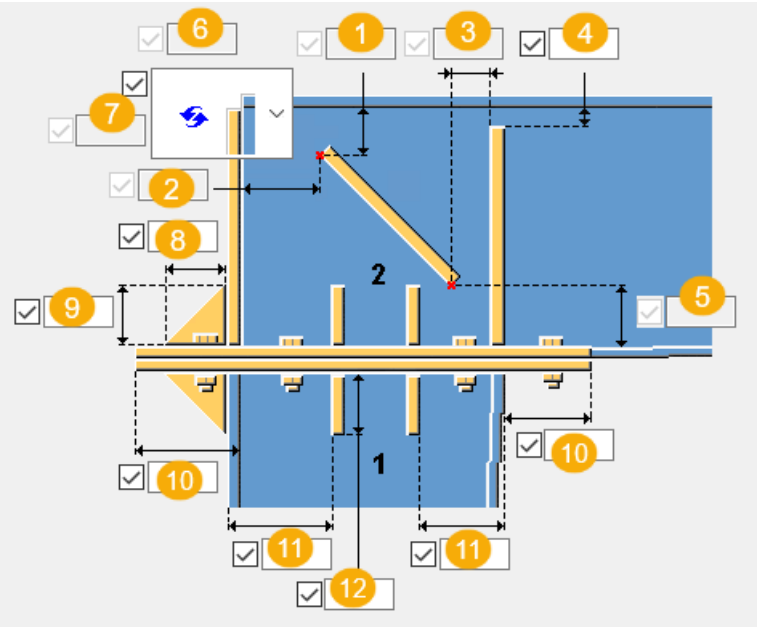
- **Horizontal**
- **Perpendicular**
- **Vertical**

Create web plate or diagonal stiffener

Option	Description
	Default Web plate is created. AutoDefaults can change this option.
	Diagonal stiffener is created.

Option	Description
	Web plate is created.

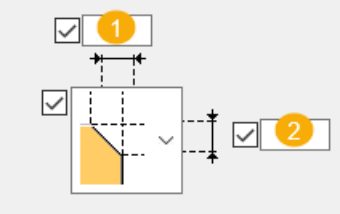
Dimensions



	Description
1	Vertical distance of the diagonal stiffener upper input point from the secondary part web.
2	Horizontal distance of the diagonal stiffener upper input point from the secondary part flange.
3	Horizontal distance of the diagonal stiffener lower input point from the knee stiffener.
4	Knee stiffener edge distance from the secondary part web.
5	Vertical distance of the diagonal stiffener lower input point from the secondary part web.
6	Horizontal dimension of the corner lap stiffener.
7	Vertical dimension of the corner lap stiffener. If the corner lap stiffener is not created, this is the knee end plate dimension from the secondary part flange.

	Description
8	Horizontal dimension of the external stiffener.
9	Vertical dimension of the external stiffener.
10	Dimension of the end plate extension from the main part web.
11	Web stiffener edge distance from the main part flange.
12	Web stiffener dimension on the main part side.

Chamfer dimensions



	Description
1	Horizontal chamfer dimension
2	Vertical chamfer dimension

Chamfer shape

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex chamfer
	Concave chamfer

Parts tab

Use the **Parts** tab to define the properties of plates and stiffeners.

Parts

Option	Description
End plate	Thickness and width of the end plate
Diagonal stiffener	Thickness and width of the diagonal stiffener
Web plate	Thickness of the web plate
Web stiffener	Thickness and width of the web stiffeners
External stiffener	Thickness of the external stiffener
Internal stiffener	Thickness, width and height of the internal stiffener
Knee stiffener	Thickness and width of the knee stiffener
Knee end plate	Thickness and width of the knee end plate
Corner lap stiffener	Thickness of the corner lap stiffener

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

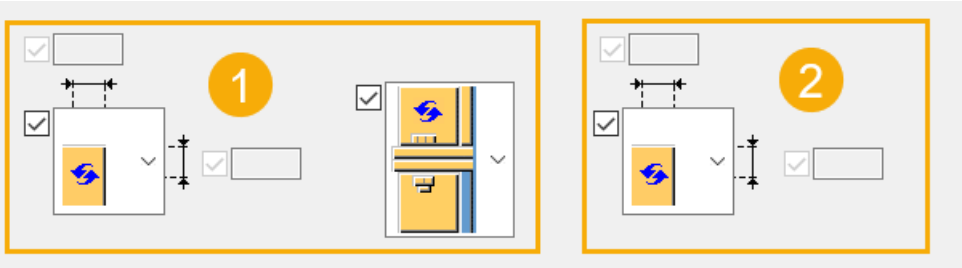
Stiffener creation

Option	Description
	Select whether bevel, square, or no flange stiffeners are created. Bevel is the default. AutoDefaults can change this option.
	Select how many web stiffeners are created, or whether they are created at all. Two stiffeners is the default. AutoDefaults can change this option.

Option	Description
	Select whether bevel, square, or no inner stiffeners are created. Bevel is the default. AutoDefaults can change this option.

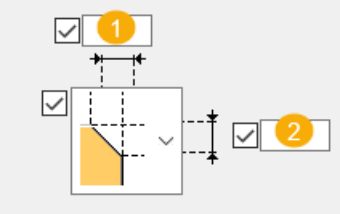
Stiffener creation

Select how many stiffeners are created, define the shape of the stiffeners, and the chamfer dimensions.



	Description
1	Define the external stiffeners.
2	Define the inner stiffeners in the corner between the main and the secondary part.

Chamfer dimensions



	Description
1	Horizontal chamfer dimension
2	Vertical chamfer dimension

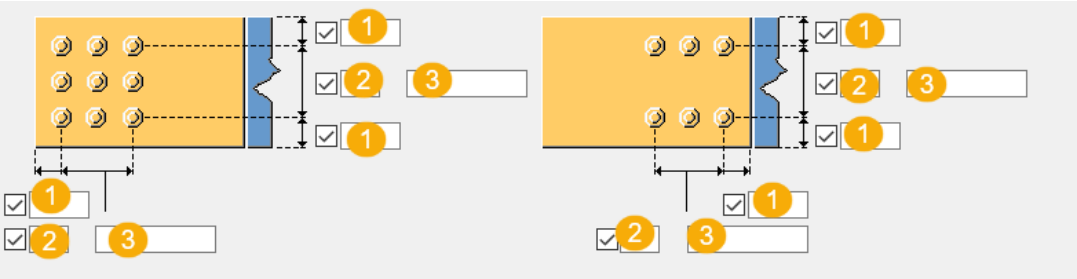
Chamfer shape

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex chamfer
	Concave chamfer

Bolts tab

Use the **Bolts** tab to define the bolt properties, number of bolts, and spacing. You can select whether outer and inner bolts are created.

Bolt group dimensions



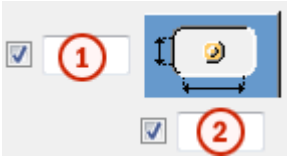
	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



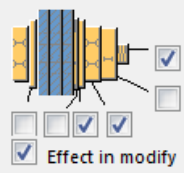
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	

Option	Description	Default
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Welds

Click the link below to find out more:

5.17 Bracing components

You can use bracing components to automatically create complete bracing connections. You can also use bracing elements (such as gusset plates and stiffeners) that you can combine with other components to create your own bracing connections

See also

Simple gusset plate connections (page 2890)

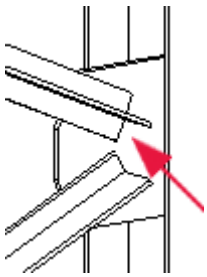
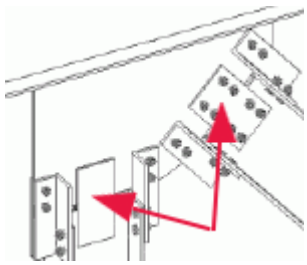
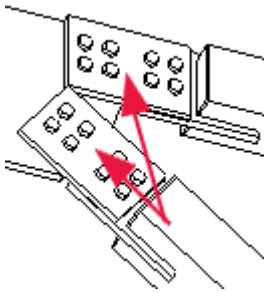
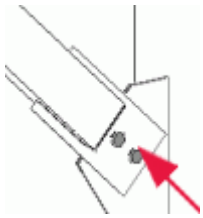
Corner bracing connections (page 3022)

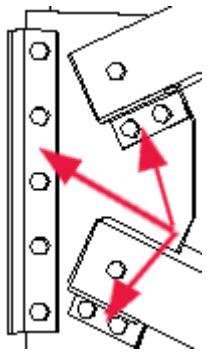
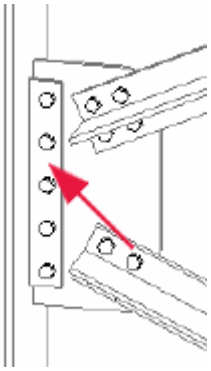
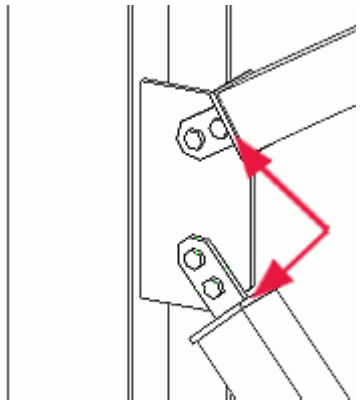
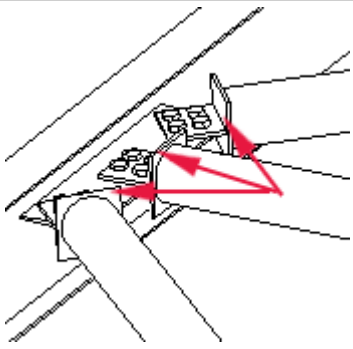
Windbracing connections (page 3146)

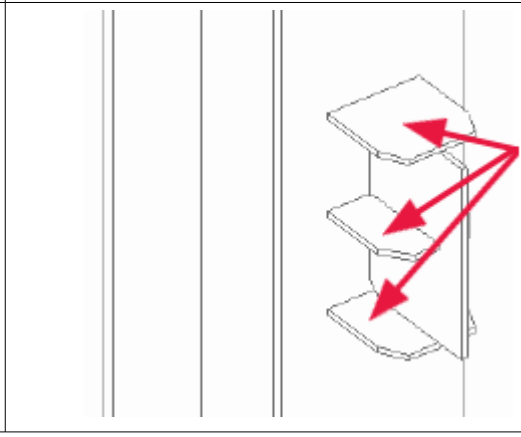
Bracing connection elements (page 3174)

Glossary of parts

Bracing components create some or all of the following parts:

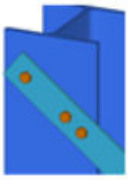
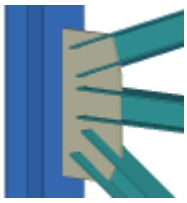
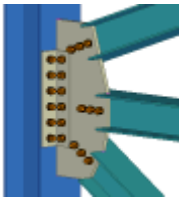
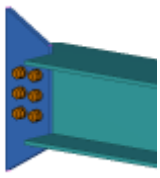
Part	Image
Gusset plate	
Connection plate	
Cover plate	
Tongue plate A *tongue plate* slots into a hollow profile brace.	

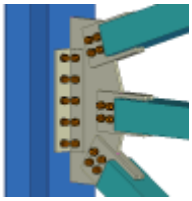
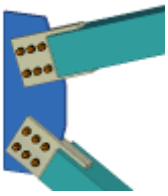
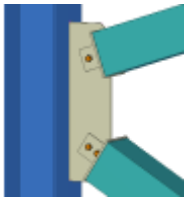
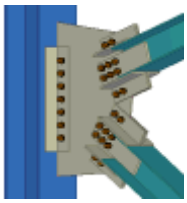
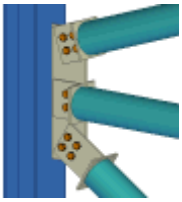
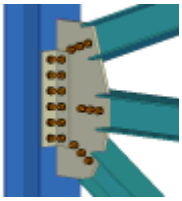
Part	Image
Clip angle	
Shear tab	
Seal plate *Seal plates* seal the ends of hollow braces. In this example they are used with chamfered connection plates.	
Tee *Tees* are formed from T or cut I profiles, or two plates. Tees seal hollow braces and connect them to another part (e.g. a gusset plate)	

Part	Image
Stiffener	

Simple gusset plate connections

Simple gusset plate connections automatically connect braces to a single main part, using a gusset plate. Braces either connect directly to the gusset plate, or use connection material. Tekla Structures includes the following simple gusset plate connections:

Component	Image	Description
Simple bolt connection (5) (page 2891)		Creates a bolted connection between two steel profiles.
Welded gusset (10) (page 2896)		Welds braces to a gusset plate.
Bolted gusset (11) (page 2904)		Bolts braces to a gusset plate. Optional clip angles or shear tabs.
Bracing cross (19) (page 2929)		Bolts twin-profile braces to an existing gusset plate.

Component	Image	Description
Tube gusset (20) (page 1992)		Bolts braces to a gusset plate using connection plates and optional tongue plates. Seals braces.
Tube crossing (22) (page 2936)		Bolts braces to an existing gusset plate using connection plates and optional tongue plates. Seals braces.
Std bracing connection (67) (page 2948)		Bolts 1 or 2 braces to a gusset plate using connection plates. Seals hollow braces.
Wrapped cross (61) (page 2956)		Bolts braces to an existing gusset plate, using connection plates and clip angles.
Gusseted cross (62) (page 2968)		Bolts braces to a gusset plate using clip angles and connection plates. Welds or bolts the gusset plate to the beam or column.
Portal bracing (105) (page 2990)		Connects up to three hollow braces to a beam, using a gusset plate and tees.
Bolted gusset (196) (page 3000)		Bolts braces to a gusset plate. Optional clip angles or shear tabs.

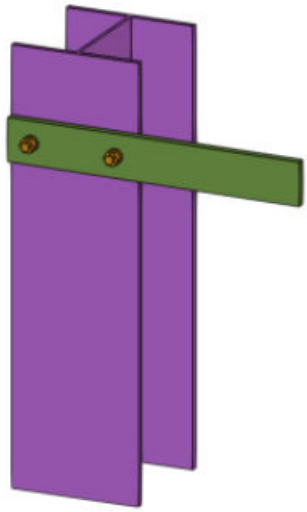
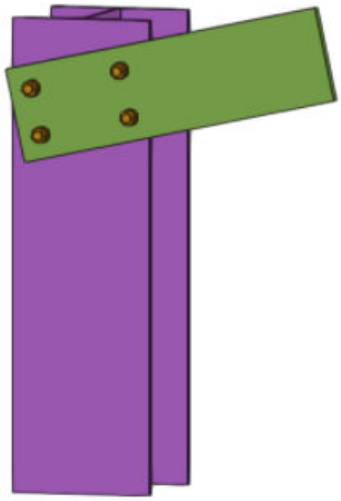
Simple bolt connection (5)

Simple bolt connection (5) creates a bolted connection between two steel profiles.

Objects created

- Bolts

Use for

Situation	Description
	Bolted connection
	Bolted connection, skewed secondary part

Before you start

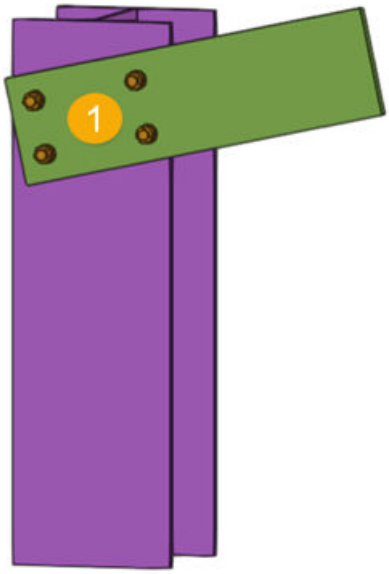
Create two steel profiles.

Selection order

1. Select the main part (first steel profile).
2. Select the secondary part (second steel profile).

   The connection is created automatically when you select the secondary part.

Part identification key

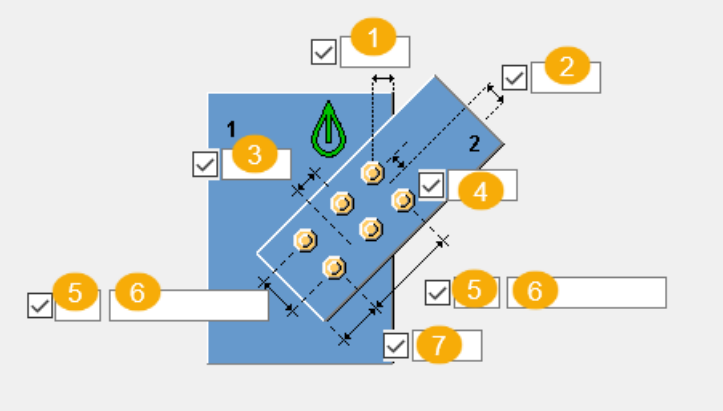


	Description
1	Bolts

Picture tab

Use the **Picture** tab to define the dimensions and position of the bolts.

Dimensions



	Description
1	Bolt edge distance from the main part edge.
2	Horizontal bolt distance from the center of the secondary part.
3	Vertical bolt distance from the center of the secondary part.
4	Bolt offset from the line defined by the horizontal bolt distance (**2**).
5	Number of bolts.
6	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
7	Horizontal bolt distance from the edge of the secondary part.

Bolts tab

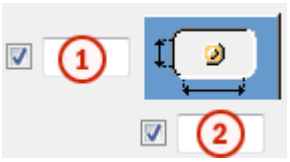
Use the **Bolts** tab to define the bolt properties.

Basic bolt properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog. `20 mm`
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog. `7990`
Tolerance	Gap between the bolt and the hole.	`3 mm`
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Assembly type	Location where the bolts should be attached.	`Site`
Rotation	Defines the bolt rotation.	`Back`

Slotted holes

You can define slotted, oversized, or tapped holes.

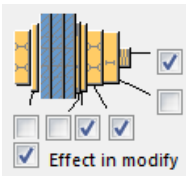


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

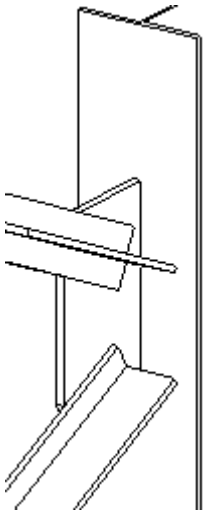
Welded gusset (10)

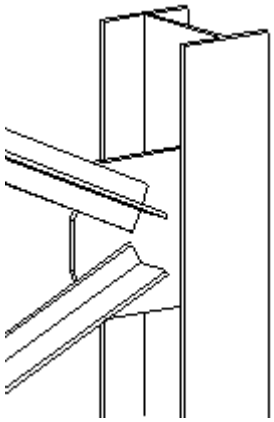
Welded gusset (10) connects 1 to 10 braces to a beam or column using a gusset plate that is welded to the web or flange of the beam or column. Braces are welded to the gusset plate.

Objects created

- Gusset plate
- Seal plates (hollow braces)
- Stiffeners
- Welds
- Cuts

Use for

Situation	Description
	Brace profile: T Gusset plate is welded to the column flange. Braces are welded to the gusset plate.

Situation	Description
	Brace profile: T Gusset plate is welded to the column web. Braces are welded to the gusset plate.

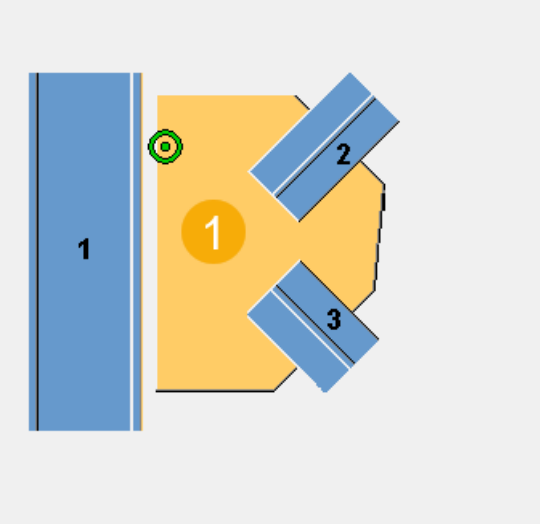
Before you start

Create a beam or a column, and 1 to 10 braces.

Selection order

1. Select the main part (beam or column).
2. Select the secondary part (first brace).
3. Select the second secondary part (second brace).
4. Select the subsequent secondary parts (subsequent braces).
5. Click the middle mouse button to create the component.

Part identification key

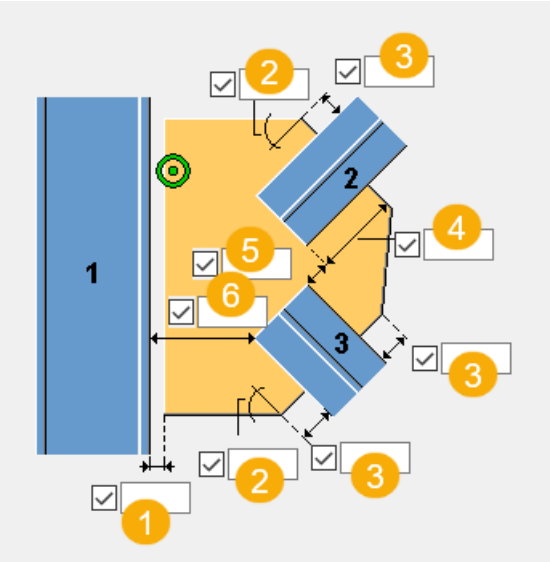


	Description
1	Gusset plate

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab
Use the **Picture** tab to define the dimensions that control the position and shape of the gusset plate.

Gusset plate dimensions



	Description
1	Define the gap distance between the gusset plate edge and the main part.
2	Define the gusset plate corner angle (in degrees). This value affects the gusset plate shape.
3	Define the length of the edge of the gusset plate. This value affects the gusset plate shape.
4	Define the brace length on the gusset plate.
5	Define the distance between the braces.
6	Define the distance between the main part and the brace.

Gusset tab
Use the **Gusset** tab to define the gusset plate properties.

Gusset plate

Part	Description
Gusset	Thickness, width and height of the gusset plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

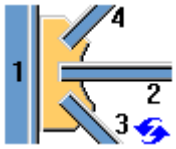
NOTE The following examples show only some of the available options. You will find more options on the **Gusset** tab.

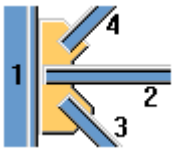
Gusset plate shape

When you select the option to optimize the gusset plate weight, you can define whether the selection order of the braces affects the position of the braces.

Option	Description
	Default AutoDefaults can change this option.
	This option optimizes the gusset plate weight.

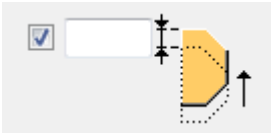
Brace position

Option	Description
	Default The brace position is not affected. AutoDefaults can change this option.

Option	Description
	The first selected brace is placed closest to the main part.

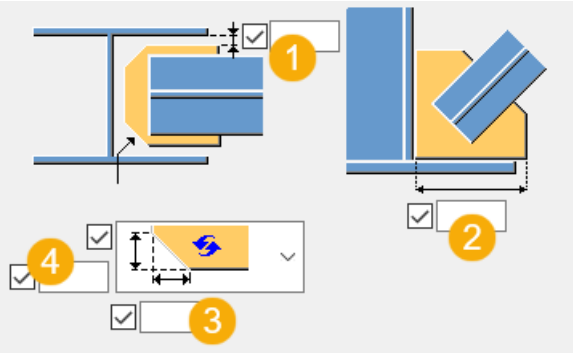
Gusset plate position on the brace

Define where to place the gusset plate on the brace. If needed, you can fine-tune the gusset plate position by moving the gusset plate in the z or in the y direction.

Option	Description
	Default Gusset plate is positioned in the middle of the brace. AutoDefaults can change this option.
	Gusset plate is positioned on the top flange of the brace.
	Define how much the gusset plate is moved in the z direction.
	Define how much the gusset plate is moved in the y direction.

Gusset plate chamfer

Define the gusset plate chamfer type and dimensions.

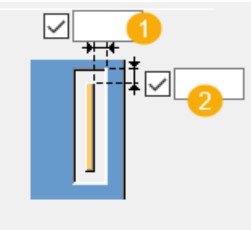


	Description
1	Distance between the gusset plate and the inner flange of the main part.

	Description
2	Horizontal distance between the gusset plate edge and the flange of the main part.
3	Horizontal dimension of the chamfer.
4	Vertical dimension of the chamfer.

Cut size

If the gusset plate runs through the main part, define the size of the cut created for the gusset plate.



	Description
1	Define the horizontal size of the cut.
2	Define the vertical size of the cut.

Brace conn tab

Use the **Brace conn** tab to define the seal plate, brace notch and slot properties.

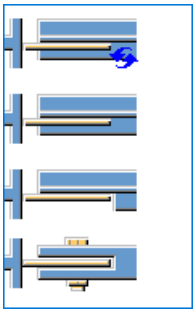
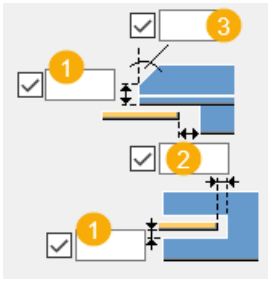
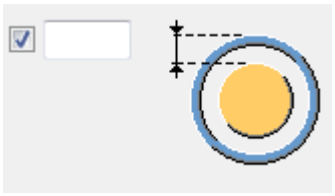
Seal plate

Part	Description
Seal plate	Thickness of the seal plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in

Option	Description	Default
		File menu --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Brace notch

Option	Description
	Select whether the brace is notched.
	1. Vertical notch dimension. 2. Horizontal notch dimension. 3. Notch angle.
	Define the plate distance from the brace outer edge.

Stiffeners tab

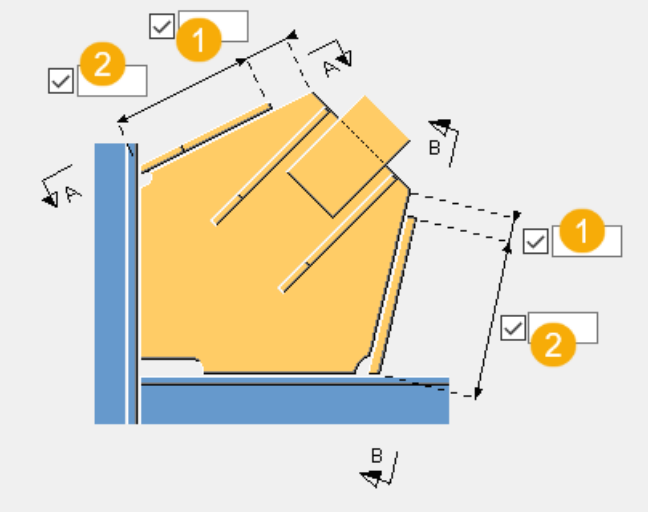
Use the **Stiffeners** tab to define the stiffener properties and dimensions.

Stiffeners

Part	Description
Stiffener 1, **Stiffener 2**	Define the stiffener thickness.

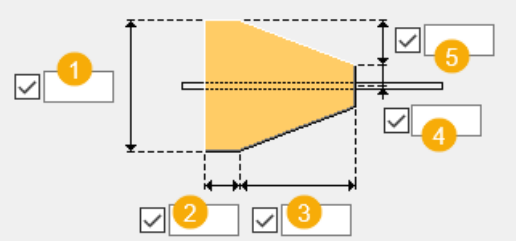
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Stiffener length



1	Distance between the stiffener edge and the gusset plate edge.
2	Length of the stiffener.

Stiffener dimensions



1	Width of the stiffener.
2	Length of the stiffener base.
3	Length of the skew part of the stiffener.
4	Distance from the stiffener center line.
5	Vertical distance between the stiffener base and the skew part.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

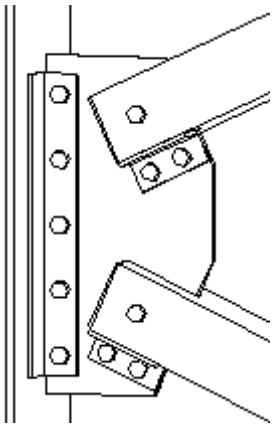
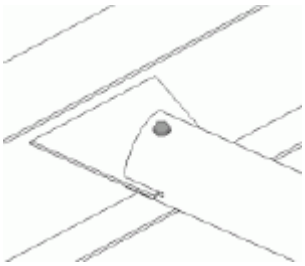
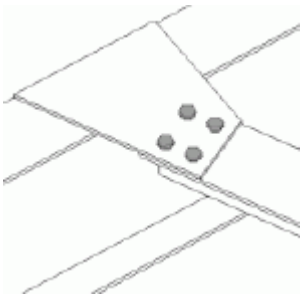
Bolted gusset (11)

Bolted gusset (11) connects 1 to 10 braces to a beam or column using a gusset plate, which is bolted or welded to the beam or column. The braces are bolted to the gusset plate. Clip angles can be created either at the ends of the braces, or on each side.

Objects created

- Gusset plate
- Clip angles or shear tabs (optional) that connect the gusset plate to the beam or column
- Clip angles (optional) that connect the brace to the gusset plate
- Connection plates
- Seal plates (hollow braces)
- Stiffeners (optional)
- Bolts
- Welds
- Cuts

Use for

Situation	Description
	Brace profile: RHS Gusset plate is bolted to the beam flange using a clip angle. Braces are slotted around the gusset plate and attached to it using bolts and clip angles.
	Brace profile: Tube Gusset plate is welded to the beam web. Brace is notched around the gusset plate and pinned to it.
	Brace profile: T Gusset plate is welded to the beam flange. Brace is bolted to the gusset plate.

Situation	Description
	Brace profile: L Gusset plate is welded to the column flange. Brace is bolted to the gusset plate.

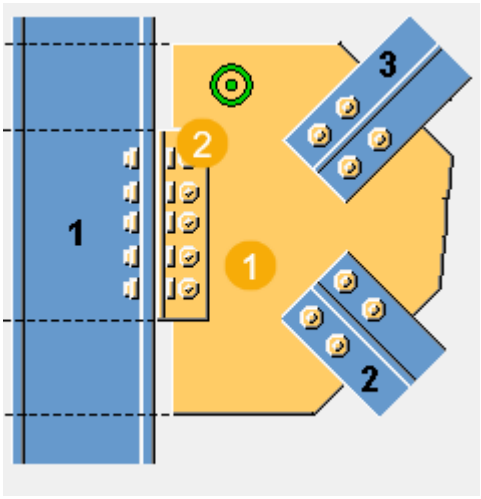
Before you start

Create a beam or column, and 1 to 10 braces.

Selection order

1. Select the main part (beam or column).
2. Select the secondary part (first brace).
3. Select the second secondary part (second brace).
4. Select the subsequent secondary parts (subsequent braces).
5. Click the middle mouse button to create the component.

Part identification key

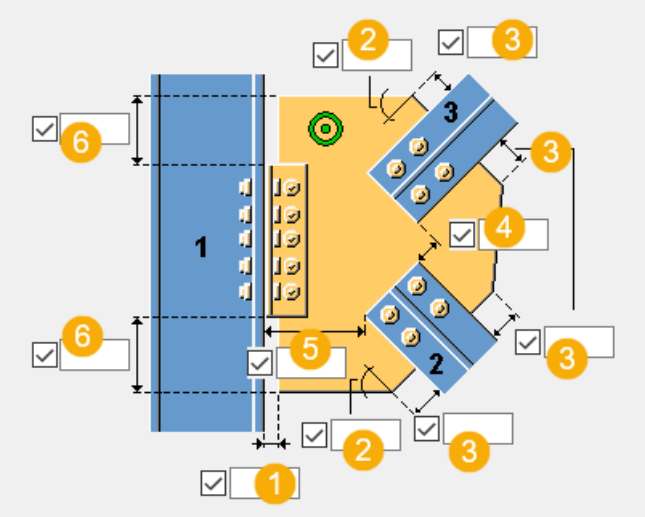


	Description
1	Gusset plate
2	Clip angle

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab
Use the **Picture** tab to define the dimensions that control the position and shape of the gusset plate.

Dimensions



	Description
1	Define the gap distance between the gusset plate edge and the main part.
2	Define the gusset plate corner angle (in degrees). This value affects the gusset plate shape.
3	Define the length of the edge of the gusset plate. This value affects the gusset plate shape.
4	Define the distance between the braces.
5	Define the distance between the main part and the brace.
6	Define the distance between the clip angle or connection plate edge and the gusset plate edge.

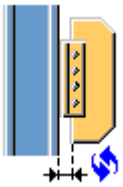
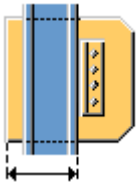
NOTE The following examples show only some of the available options. You will find more options on the **Picture** tab.

Gusset plate positioning

Define how the gusset plate is positioned when a base plate is used.

Option	Description
	Default Gusset plate is parallel to the main part. AutoDefaults can change this option.
	Gusset plate is parallel to the brace.

Gusset plate dimensions

Option	Description
	Default Gusset plate does not run through the main part. Define the gusset plate cut dimension. AutoDefaults can change this option.
	Gusset plate does not run through the main part. Define the gusset plate cut dimension.
	Gusset plate runs through the main part. Define the gusset plate extension dimension.

Gusset tab

Use the **Gusset** tab to control the gusset plate properties, shape and position, and clip angle properties and orientation.

Plates

Option	Description
Gusset	Thickness, width and height of the gusset plate.
Connection plates	Thickness and width of the connection plate.
L profile	Select the clip angle profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

NOTE The following examples show only some of the available options. You will find more options on the **Gusset** tab.

Gusset plate connection

Define how the gusset plate is connected to the main part.

Option	Description
	Default Gusset plate is welded directly to the main part. AutoDefaults can change this option.
	Gusset plate is connected to the main part with clip angles. Select to which side of the gusset plate the clip angles are created.
	Gusset plate is connected to the main part with a connection plate. Select to which side of the gusset plate the connection plate is created.

Clip angle orientation

Define how the clip angle is placed on the connection.

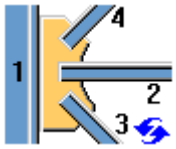
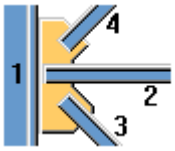
Option	Description
	Default Clip angle is placed on the connection so that the longer leg is connected to the gusset plate. AutoDefaults can change this option.
	Clip angle is placed on the connection so that the longer leg is connected to the main part.

Gusset plate shape

When you select the option to optimize the gusset plate weight, you can define whether the selection order of the braces affects the position of the braces.

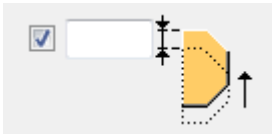
Option	Description
	Default AutoDefaults can change this option.
	This option optimizes the gusset plate weight.

Brace position

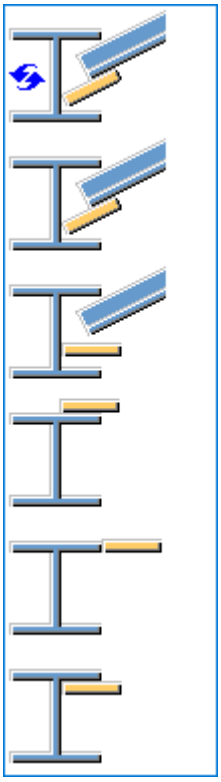
Option	Description
	Default The brace position is not affected. AutoDefaults can change this option.
	The first selected brace is placed closest to the main part.

Gusset plate position on the brace

Define where to place the gusset plate on the brace. If needed, you can fine-tune the gusset plate position by moving the gusset plate in the z or in the y direction.

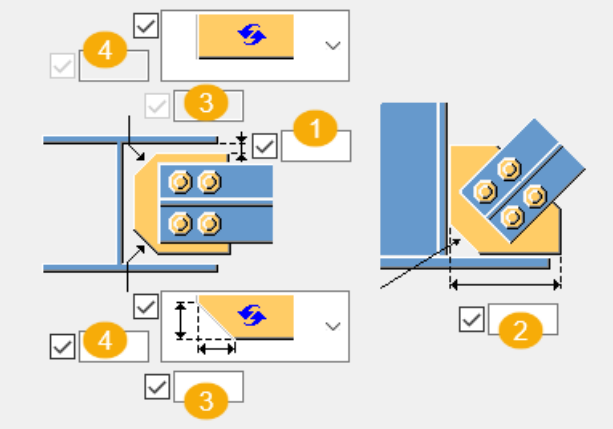
Option	Description
	Default Gusset plate is positioned in the middle of the brace. AutoDefaults can change this option.
	Gusset plate is positioned on the top flange of the brace.
	Define how much the gusset plate is moved in the z direction.
	Define how much the gusset plate is moved in the y direction.

Gusset plate position on the column or beam

Option	Description
	Select the gusset position. The default option is parallel to the brace.

Gusset plate chamfer

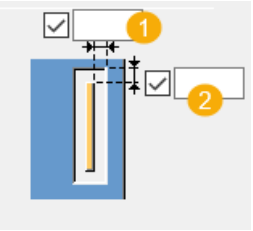
Define the gusset plate chamfer type and dimensions, and the angle limit for parallel bracing.



	Description
1	Distance between the connection plate and the inner flange of the main part.
2	Horizontal distance between the gusset plate edge and the flange of the main part.
3	Horizontal dimension of the chamfer. By default, the second chamfer is not created.
4	Vertical dimension of the chamfer. By default, the second chamfer is not created.

Cut size

If the gusset plate runs through the main part, define the size of the cut created for the gusset plate.



	Description
1	Define the horizontal size of the cut.
2	Define the vertical size of the cut.

Brace conn

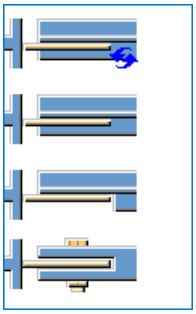
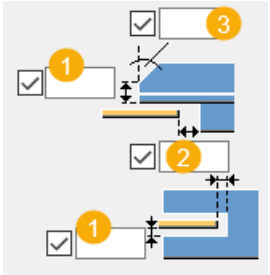
Use the **Brace conn** tab to define the seal plate, brace notch and slot properties.

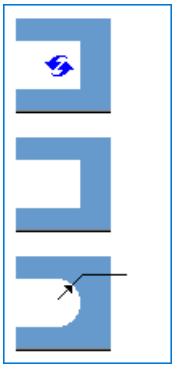
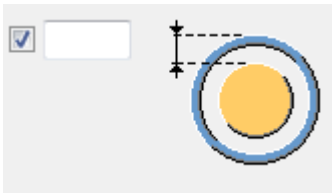
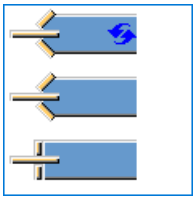
Seal plate

Part	Description
Seal plate	Thickness of the seal plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Brace notch

Option	Description
	Select whether the brace is notched.
	1. Vertical notch dimension. 2. Horizontal notch dimension. 3. Notch angle.

Option	Description
	Select the brace notch shape. Define the radius for the round shape.
	Define the plate distance from the brace outer edge.
	Select whether to create bevel cuts at the brace ends or whether to create a square brace end.

Stiffeners tab
Use the **Stiffeners** tab to define the stiffener properties and dimensions.

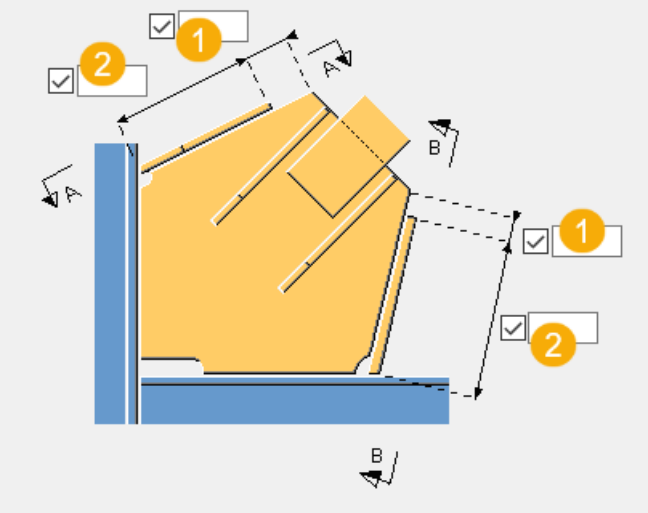
Parts

Part	Description
Stiffener 1, **Stiffener 2**	Thickness of the stiffener.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in

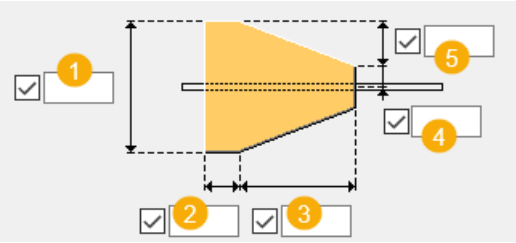
Option	Description	Default
		File menu --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Stiffener length



1	Define the distance between the stiffener edge and the gusset plate edge.
2	Define the length of stiffener.

Stiffener dimensions



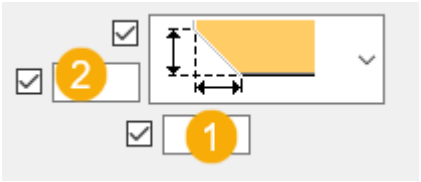
1	Define the width of the stiffener.
2	Define the length of the stiffener base.
3	Define the length of the skew part of the stiffener.
4	Define the distance from the stiffener center line.

5	Define the vertical distance between the stiffener base and the skew part.

Chamfer type

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Chamfer dimensions

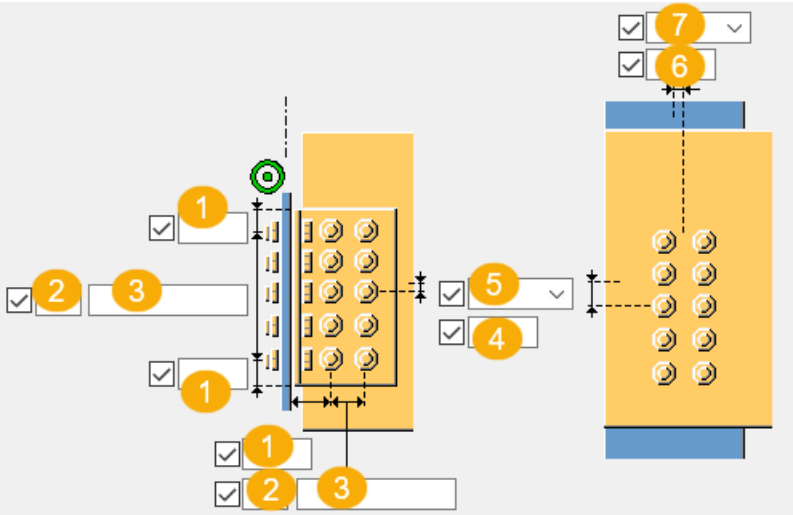


	Description
1	Horizontal dimension of the chamfer.
2	Vertical dimension of the chamfer.

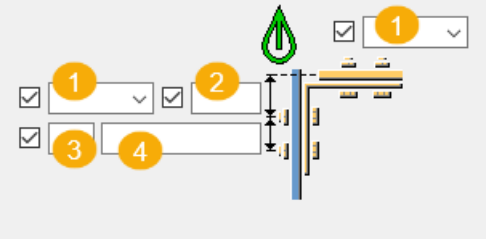
Gusset bolts tab

Use the **Gusset bolts** tab to control the bolt group properties for bolts that connect the gusset plate to the main part, and to control the clip angle attachment.

Bolt group dimensions on gusset plate



1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Dimension for vertical bolt group position.
5	Select how to measure the dimensions for vertical bolt group position.
6	Dimension for horizontal bolt group position.
7	Select how to measure the dimensions for horizontal bolt group position.

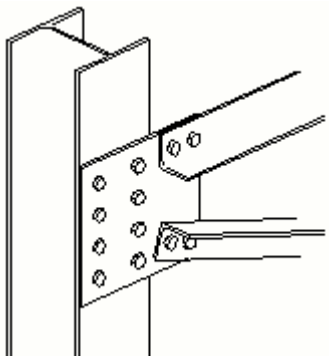


	Description
1	Location where the bolts should be attached.

	Description
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt group position

Gusset horizontal position and offset options only affect the following situation, where the gusset is connected to the column flange. When the position is middle, the bolt group offset is calculated from the center line of the column, for example.



Gusset vertical position and offset options affect the bolt group connecting the gusset directly to the flange and also the bolt groups connecting the gusset to the clip angle or connection plate.

NOTE The following examples show only some of the available options. You will find more options on the **Gusset bolts** tab.

Clip angle attachment type

Define how the clip angle is attached to the gusset plate and to the main part.

Option	Description
	Default Both parts are bolted. AutoDefaults can change this option.
	Automatic When the main part is a tube profile, the clip angles are welded to the main part and bolted to the

Option	Description
	secondary part. Otherwise the clip angles are bolted to both parts.
	Main part is bolted and secondary part is welded.
	Main part is welded and secondary is part bolted.
	Both parts are bolted.
	Both parts are welded.

Bolts on gusset plate

Define whether the gusset plate is connected to the main part with bolts when no clip angles are used.

Option	Description
	Default Bolts are not created in the gusset plate. AutoDefaults can change this option.
	Bolts are created in the gusset plate.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3

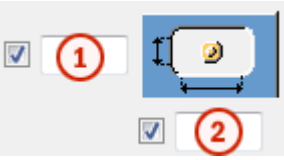
Option	Description
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.



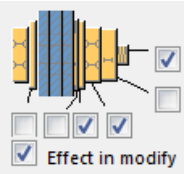
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	

Option	Description	Default
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

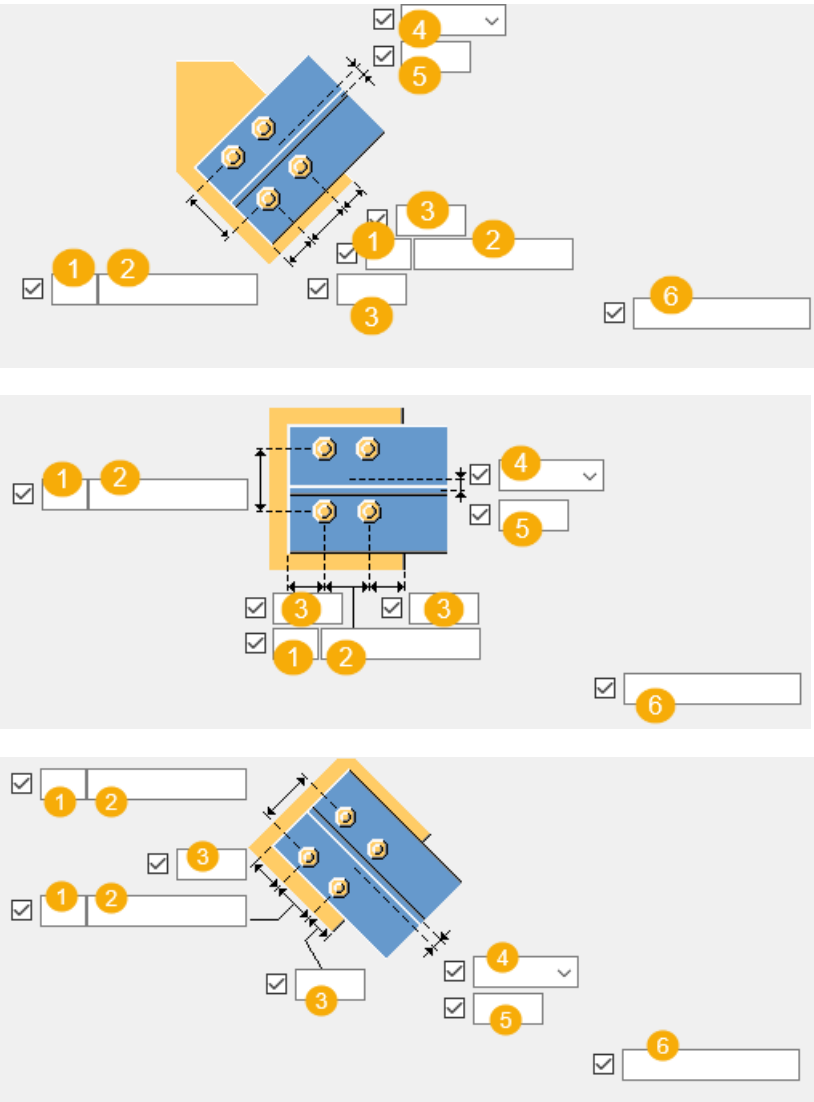
Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Bolting direction

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1
	Bolting direction 2

Brace bolts 1 / Brace bolts 2 / Brace bolts 3 tab
Use the **Brace bolts 1**, **Brace bolts 2** and **Brace bolts 3** tabs to control the bolts that connect the first, the second, and the subsequent braces to the gusset plate.

Bolt group dimensions

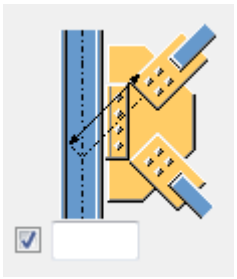
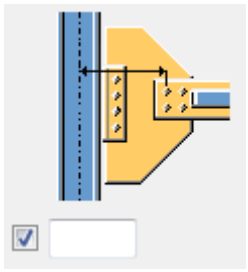
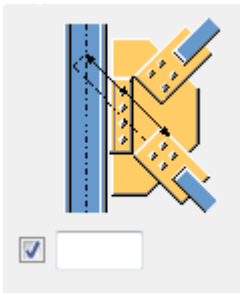


	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

	Description
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Select how to measure the dimensions for vertical bolt group position.
5	Dimension for vertical bolt group position.
6	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Bolt distance

Define the minimum distance from the connection plate bolts to the intersection point of the main part and brace center lines. If a brace is perpendicular to the main part, the distance is measured from the main part center line to the nearest bolts.

Option	Option	Option
		

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2

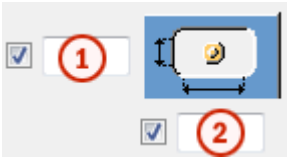
Option	Description
	Staggered type 3
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



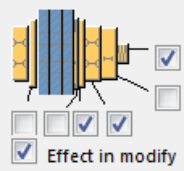
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Bolting direction

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1

Option	Description
	Bolting direction 2

Angle bolts tab
Use the **Angle bolts** tab to control the bolts that connect the clip angles.

Part

Option	Description
L profile	Select the clip angle profile from the profile catalog.

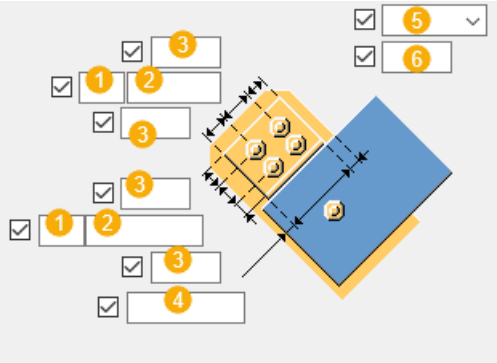
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	

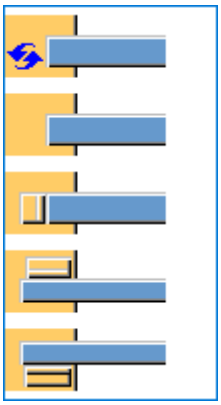
Option	Description	Default
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Bolt group dimensions



	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Define the edge distance between the clip angle and the brace.
5	Select how to measure the dimensions for vertical bolt group position.
6	Dimension for vertical bolt group position.

Clip angle position

Option	Description
	Select the clip angle position.

Bolting direction

Option	Description
	Default Bolting direction 1 AutoDefaults can change this option.
	Bolting direction 1
	Bolting direction 2

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Bracing cross (19)

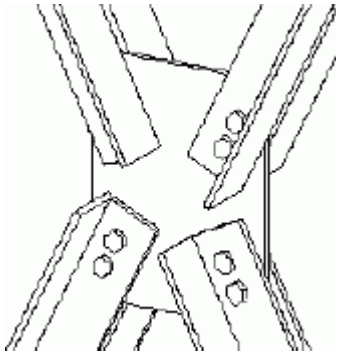
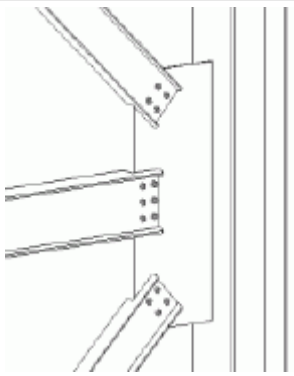
Bracing cross (19) bolts one or more twin-profile braces to an existing gusset plate.

TIP To create a gusset plate, use the Standard gusset (1065) (page 2515) component, or create a countour plate.

Objects created

- Bolts
- Cuts
- Welds

Use for

Situation	Description
	Brace profile: L Braces are bolted directly to the gusset plate.
	Brace profile: W Braces are notched around the gusset plate and bolted to the gusset plate.

Before you start

Create a gusset plate, and 1 to 10 braces.

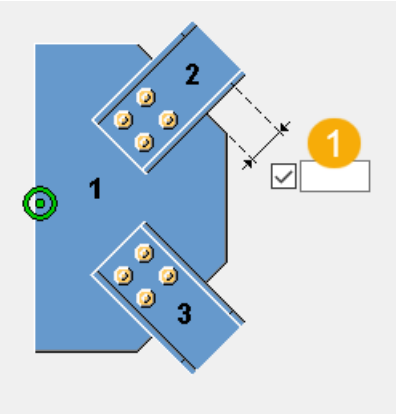
Selection order

1. Select the main part (gusset plate).
2. Select the secondary part (first brace).
3. Select the second secondary part (second brace).
4. Select the subsequent secondary parts (subsequent braces).
5. Click the middle mouse button to create the component.

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab define the flange cut dimension.

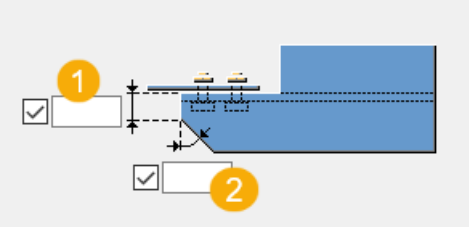


	Description
1	Define the length of the flange cut.

Parameters tab

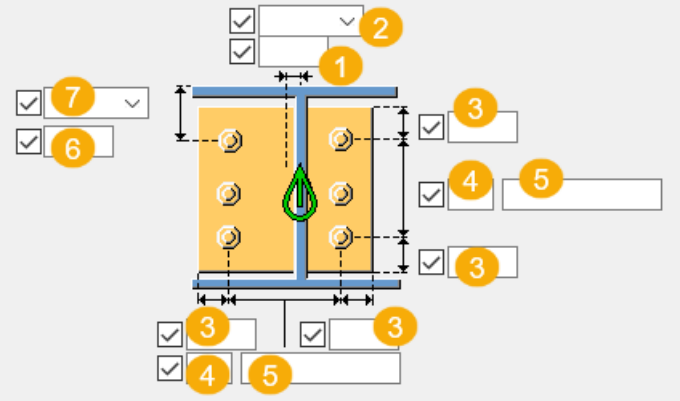
Use the **Parameters** tab to cut the brace flange.

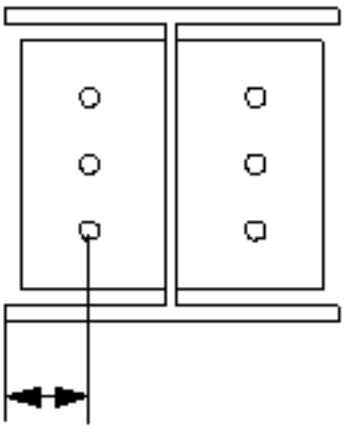
Option	Description
Fit secondary	Select whether the secondary part is fitted. Selecting **Yes** fits the secondary part according to the bolt distances.

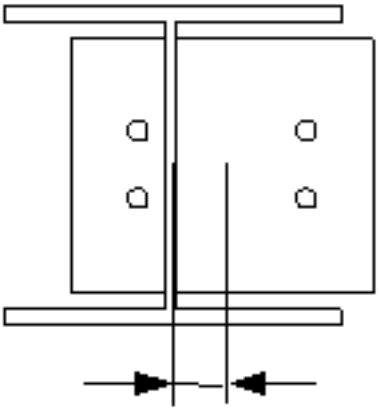
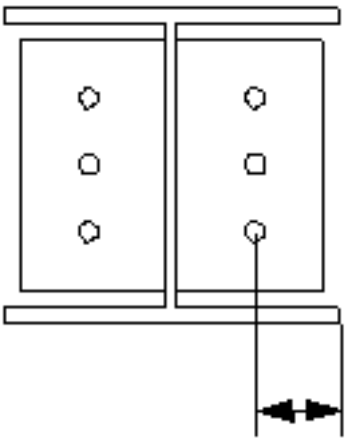
Option	Description
Cut other side flanges 	Select whether to create a triangular cut on the opposite side of the flange. 1. Vertical cut dimension. 2. Cut angle.

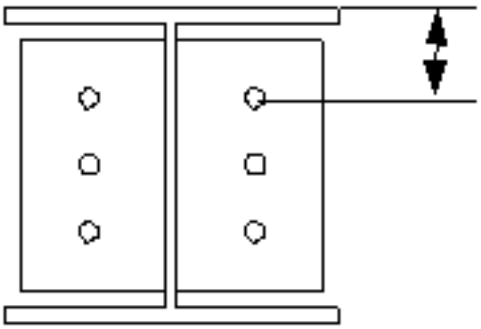
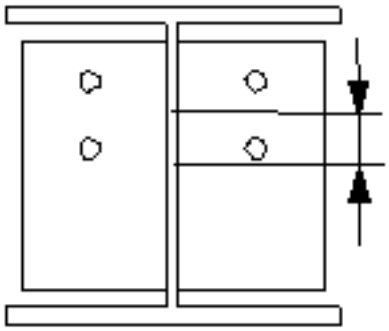
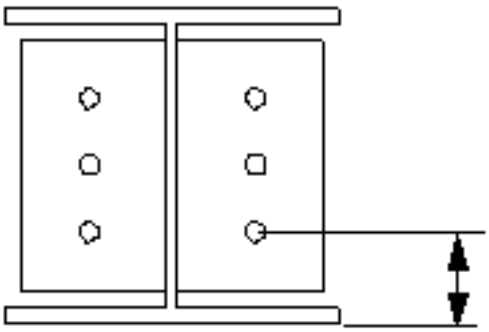
Bolts tab
Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions



	Description
1	Dimension for horizontal bolt group position.
2	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt. 

	Description
	• **Middle**: From the center line of the secondary part to the center line of the bolts.  • **Right**: From the right edge of the secondary part to the rightmost bolt. 
3	Bolt edge distance.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
6	Dimension for vertical bolt group position.

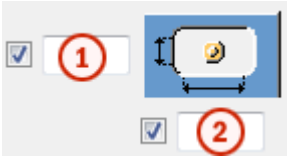
	Description
7	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



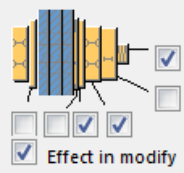
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	

Option	Description	Default
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3

Option	Description
	Staggered type 4

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Tube crossing (22)

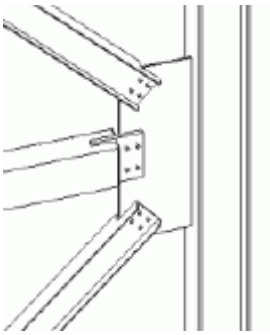
Tube crossing (22) bolts one or more braces to an existing gusset plate, directly, or using connection plates. Seals braces.

To create a gusset plate, use the Standard gusset (1065) (page 2515) component or the command for creating a contour plate.

Objects created

- Connection plates
- End plates
- Tongue plates
- Cover plates
- Bolts
- Welds

Use for

Situation	Description
	Brace profile: RHS, W RHS brace is bolted to the gusset plate using a tongue plate. W profile braces are bolted directly to the gusset plate.

Before you start

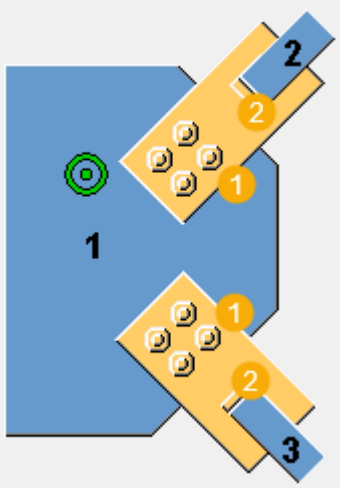
Create a gusset plate, and 1 to 10 braces.

Selection order

1. Select the main part (gusset plate).
2. Select the secondary part (first brace).
3. Select the second secondary part (second brace).
4. Select the subsequent secondary parts (subsequent braces).
5. Click the middle mouse button to create the connection.

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Part identification key

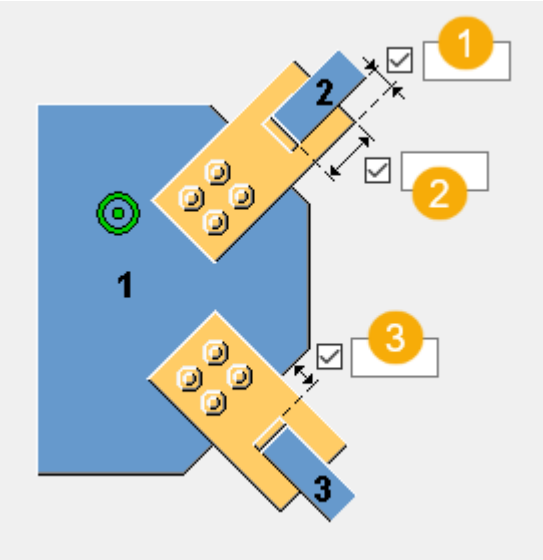


	Description
1	Connection plate

	Description
2	End plate

Picture tab
Use the **Picture** tab to define the plate and brace dimensions.

Dimensions



	Description
1	Define the length of the edge of the connection plate.
2	Define the brace length on the connection plate.
3	Define the end plate edge distance from the gusset plate.

Brace conn tab
Use the **Brace conn** tab to control the connection plate, end plate, tongue plate, and cover plate properties.

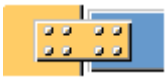
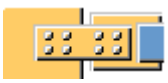
Brace connection

Option	Description
Connection plate	Thickness, width and height of the connection plate.
End plates	Thickness, width and height of the end plate.
Tongue plate	Thickness of the tongue plate.
Cover plate	Thickness, width and height of the cover plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

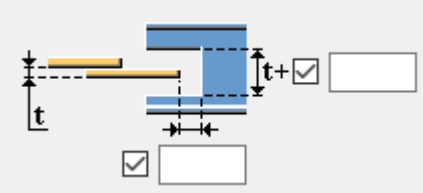
Brace connection types

Define how the brace is connected to the connection plate.

Option	Descripion
	Default Brace is welded AutoDefaults can change this option.
	Brace is welded.
	Brace is bolted.
	Brace is welded and notched around the nuts.
	Tongue plate and cover plate are created.

Cut in bracing

If needed, you can create a cut in the bracing.



Define the width of the cut in the bracing, where **t** is the thickness of the connection plate.

Define the length of the cut in the bracing from the edge of the connection plate.

Gusset and connection plate distance

Define the distance between the gusset plate and the connection plate.

Round cut in bracing

If needed, you can create a round cut in the bracing. Enter the radius value.

NOTE The following examples show only some of the available options. You will find more options on the **Brace conn** tab.

Connection plate

Define whether the brace is notched or the connection plate cut when the connection plate is connected to the brace.

Option	Description
	Default Brace is notched. AutoDefaults can change this option.
	Connection plate is cut.
	Connection plate is cut, but the part of the connection plate created inside the bracing is not deleted.

Option	Description
	If you cut the connection plate, you can define the size of the gap between the brace and the connection plate.

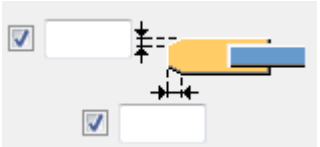
Number of connection plates

Define whether one or two connection plates are used for connecting the brace to the gusset plate.

Option	Description
	Default One connection plate. AutoDefaults can change this option.
	Two connection plates and a middle end plate at the ends of the connection plates.
	If a middle end plate is created, you can define the width of the end plate.
	Select the middle end plate position.

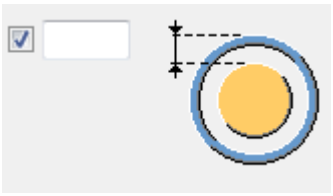
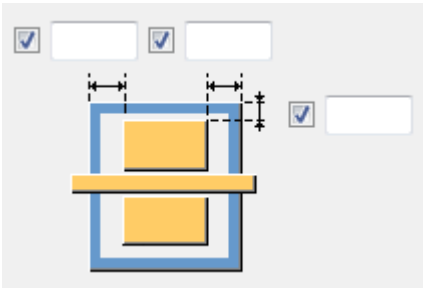
Connection plate chamfer

Define whether the connection plate is chamfered.

Option	Description
	Default No chamfers are created. AutoDefaults can change this option.
	Chamfers are created.
	If you create chamfers, define the vertical and horizontal chamfer dimensions.

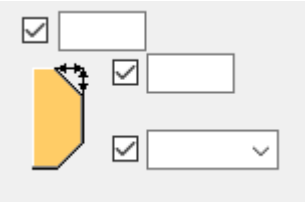
End plates

If you use the end plates to seal the braces, define the end plate shape and dimensions.

Option	Description
	Default Square end plate. AutoDefaults can change this option.
	Round end plate.
 	End plate edge distance from the brace outer edge.

End plate chamfer

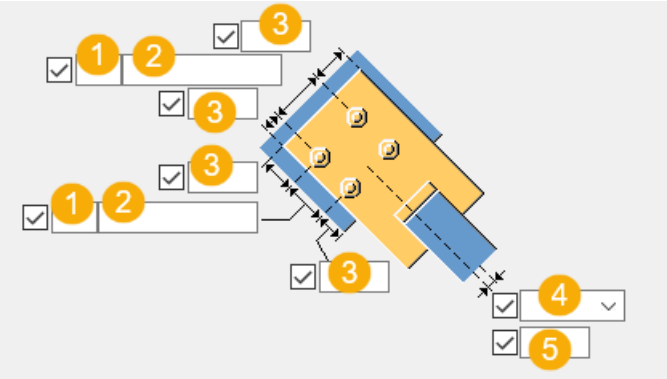
Define the end plate chamfer type and horizontal and vertical dimensions.



Brace bolts 1 tab

Use the **Brace bolts 1** tab to control the bolts that connect the braces to the gusset plate.

Bolt group dimensions



	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Select how to measure the dimensions for vertical bolt group position.
5	Dimension for vertical bolt group position.

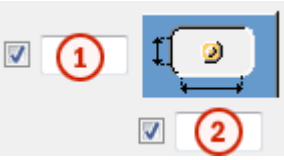
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

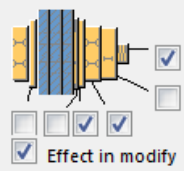


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Rib plates tab

Use the **Rib plates** tab to control the cross plate and the cover plate properties and position.

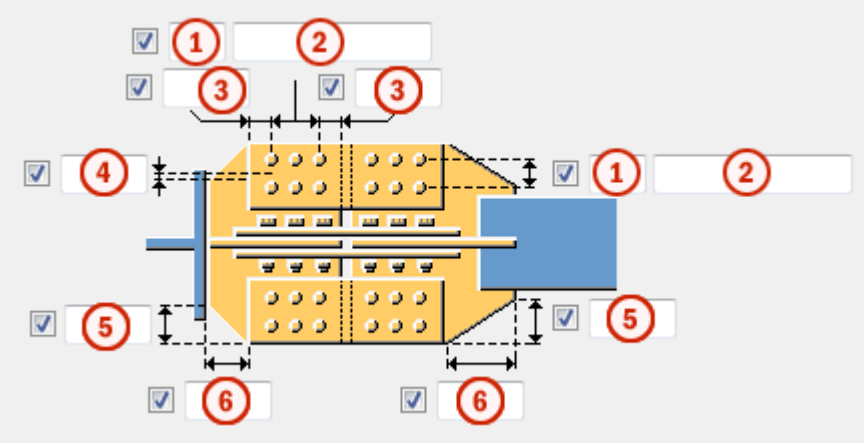
Plates

Option	Description
Cross plate	Thickness, width and height of the cross plate.

Option	Description
Cover plate	Thickness, width and height of the cover plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Bolt group dimensions



1	Number of bolts.

2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Dimension for the horizontal bolt group position.
5	Vertical dimension of the chamfer.
6	Horizontal dimension of the chamfer.

Cross and cover plate position

Option	Description
	Default Cover plate is created on both sides of the cross plate. AutoDefaults can change this option.
	Cover plate is created on both sides of the cross plate.
	Cover plate is created on the bottom of the cross plate.
	Cover plate is created on top of the cross plate.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

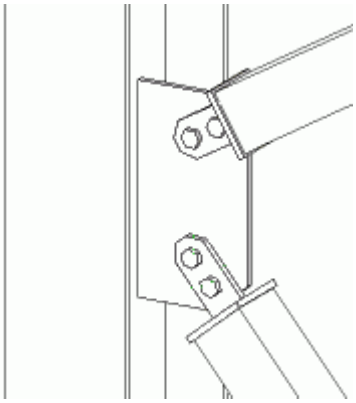
Std bracing connection (67)

Std bracing connection (67) connects 1 or 2 braces to a beam or column using a gusset plate and connection plates. Seals hollow and tube profile braces. You can define different properties for the upper and lower brace connections.

Objects created

- Gusset plate
- Connection plates
- Seal plates
- Bolts
- Welds

Use for

Situation	Description
	Brace profile: RHS Gusset plate is welded to the column web. Brace is bolted to the gusset plate using chamfered connection plates.

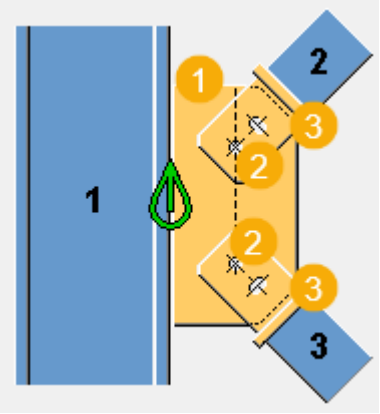
Before you start

Create a beam or column, and 1 or 2 braces.

Selection order

1. Select the main part (beam or column).
2. Select the secondary part (first brace).
3. Select the second secondary part (second brace).
4. Click the middle mouse button to create the component.

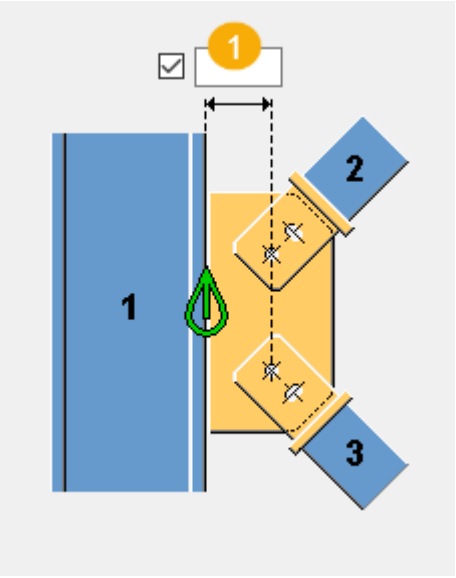
Part identification key



	Description
1	Gusset plate
2	Connection plate
3	Seal plate

Picture tab
Use the **Picture** tab to define the main part edge dimension to the bolt group.

Dimensions



	Description
1	Define the main part edge dimension to the bolt group.

Parts tab
Use the **Parts** tab to define the plate properties. You can define the plate properties for both braces.

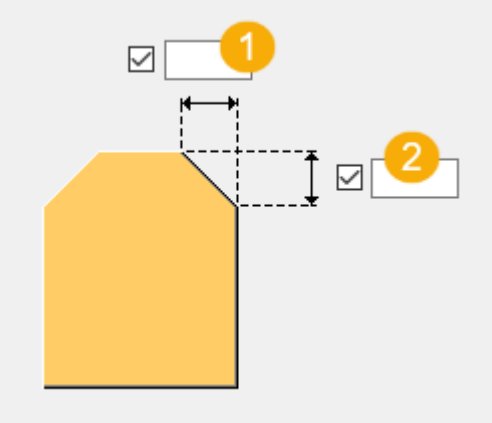
Plates

Option	Description
Connection plate	Thickness, width and height of the connection plate.
Seal plate	Thickness, width and height of the seal plate.
Seal plate profile	Select the seal plate profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab
Use the **Parameters** tab to define the gusset plate chamfer dimensions.

Chamfer dimensions

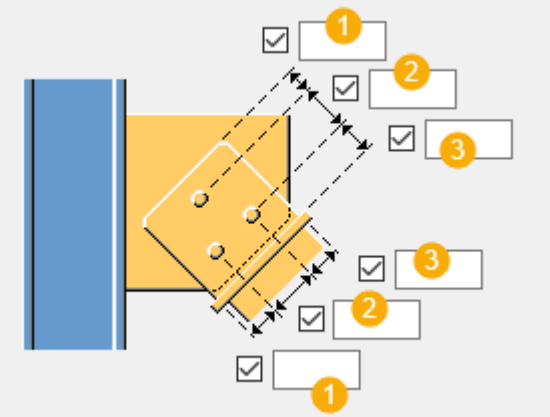


	Description
1	Horizontal chamfer dimension.
2	Vertical chamfer dimension.

Brace bolts 1 tab

Use the **Brace bolts 1** tab to define the bolt properties for the first brace.

Bolt group dimensions



	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

	Description
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.

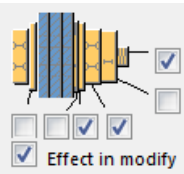
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

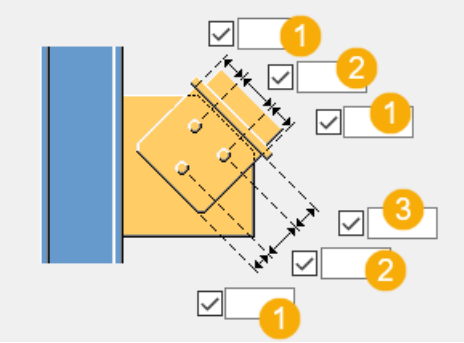
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Brace bolts 2 tab

Use the **Brace bolts 2** tab to define the bolt properties for the second brace.

Bolt group dimensions



	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within	`Yes`

Option	Description	Default
	the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Gusset tab

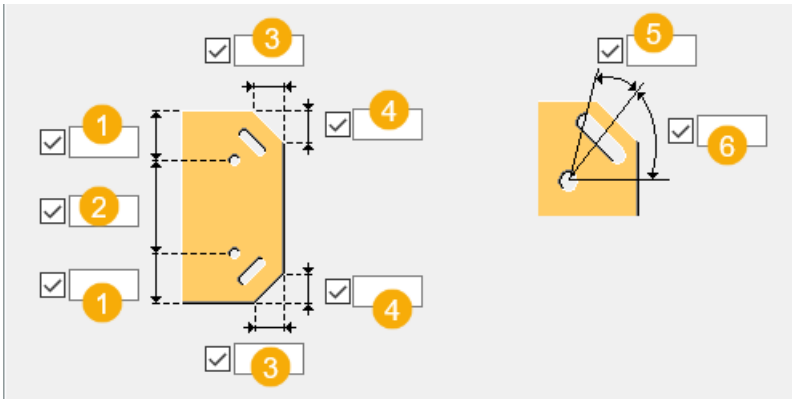
Use the **Gusset** tab to define the gusset plate dimensions and properties.

Plate

Part	Description
Gusset	Thickness, width and height of the gusset plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

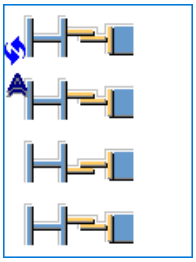
Gusset dimensions



	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Horizontal chamfer dimension.
4	Vertical chamfer dimension.
5	Upper angle of the slotted hole created in the gusset plate.
6	Lower angle of the slotted hole created in the gusset plate.

Gusset plate position

Select the gusset plate position.



General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

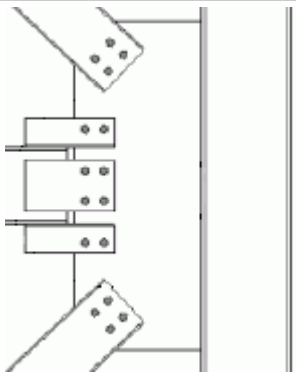
Wrapped cross (61)

Wrapped cross (61) bolts one or more braces to an existing gusset plate, using connection plates and clip angles.

Objects created

- Connection plates
- Clip angles
- Shear tabs
- Shim plates
- Bolts
- Welds

Use for

Situation	Description
	Brace profile: W Braces are bolted to the gusset plate using clip angles and connection plates.

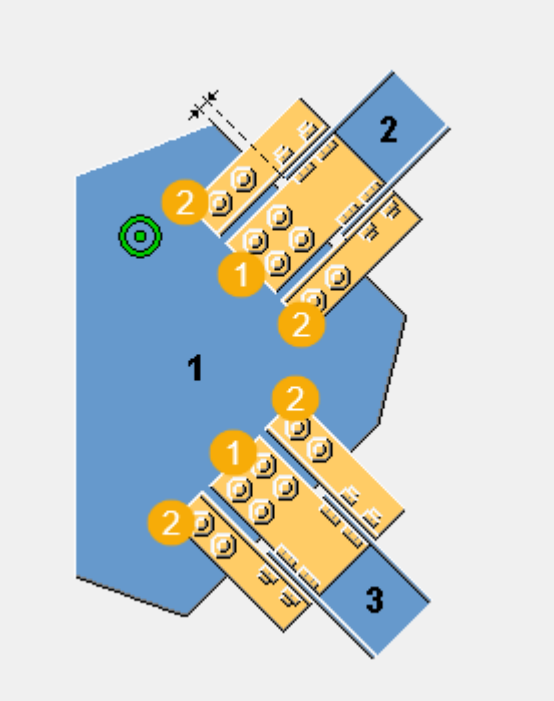
Before you start

Create a gusset plate, and 1 to 10 braces.

Selection order

1. Select the main part (gusset).
2. Select the secondary part (first brace).
3. Select the second secondary part (second brace).
4. Select the subsequent secondary part (third brace).
5. Click the middle mouse button to create the component.

Part identification key

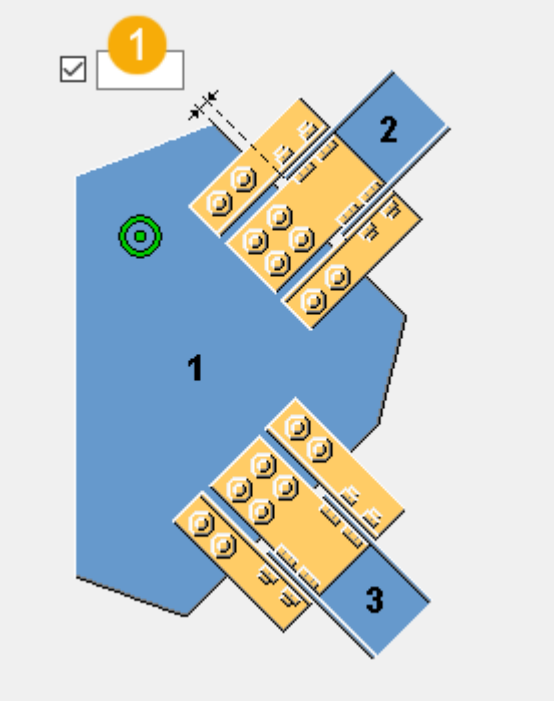


	Description
1	Connection plate
2	Clip angle

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to define the gap dimension between the gusset plate and the brace.



	Description
1	Define the dimension between the gusset plate and the brace.

Brace conn tab

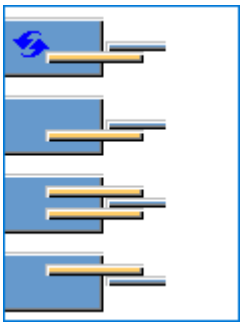
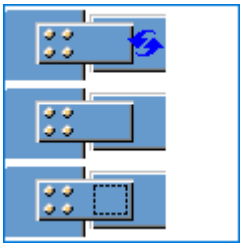
Use the **Brace conn** tab to define the connection plate, clip angle, and filler plate properties. Select whether twin profiles are used for the angle connection.

Parts

	Description
Connection plate	Thickness and width of the connection plate.
Conn. plate profile	Select the connection plate profile from the profile catalog.
Upper clip angle	Select the clip angle profile from the profile catalog.
Lower clip angle	Select the clip angle profile from the profile catalog.
Filler plate	Thickness of the filler plate.
Upper shear tab	Thickness, width, and height of the upper shear tab.
Lower shear tab	Height of the lower shear tab.

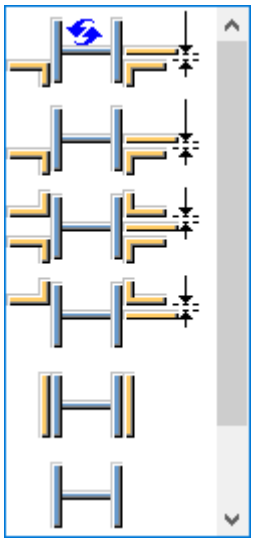
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

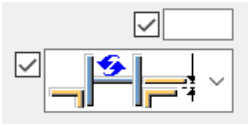
Plate creation

Option	Description
	Select whether one or two connection plates are created.
	Select whether a filler plate is created between the connection plate and the brace web. The default is that a filler plate is not created.

Clip angle creation

Define whether the braces are attached to the gusset plate using clip angles or shear tabs, and specify the number of clip angles to create. The default option is to create two clip angles below the brace web.



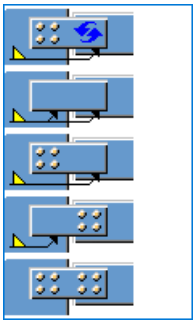


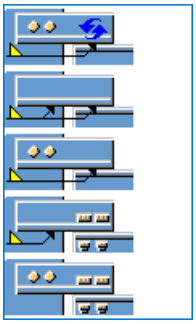
Clip angle orientation

Define how the clip angle is placed on the connection.

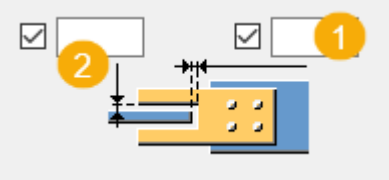
Option	Description
	Default Clip angle is placed on the connection so that the longer leg is connected to the gusset plate. AutoDefaults can change this option.
	Clip angle is placed on the connection so that the longer leg is connected to the main part.

Connection type

Option	Description
	Select the connection type (weld or bolts) between the gusset plate and the connection plate.

Option	Description
	Select the connection type (weld or bolt) between the gusset plate and the L profile.

Connection plate gap dimensions



	Description
1	Horizontal gap dimension
2	Vertical gap dimension

Shim plates tab

Use the **Shim plates** tab to define shim plate properties.

Plates

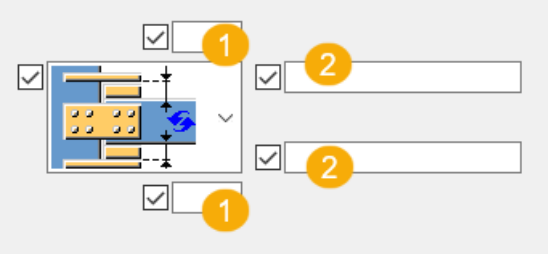
Option	Description
Shim plate 1, **Shim plate 2**, **Shim plate 3**	Thickness, width and height of the shim plates.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in

Option	Description	Default
		File menu --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Shim plate position and number

You can create shim plates when connecting braces to the gusset plate using clip angles.

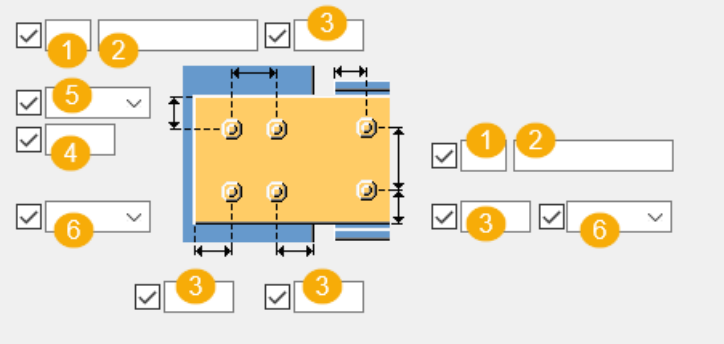


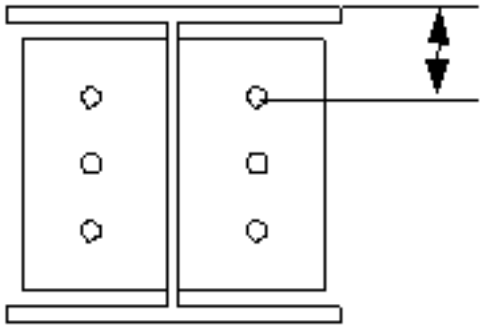
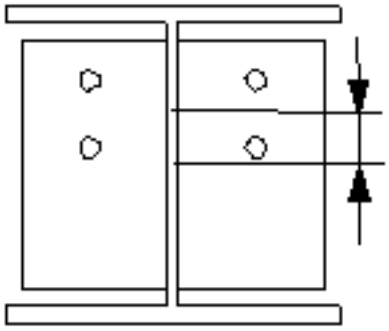
	De
1	Define the gap between the brace and connection plate.
2	Define how many shim plates are created at the top and bottom flanges. Enter the shim plate profile numbers: 1, 2 or 3. These are the numbers that are on the upper part on the **Shim plates** tab. For example, if you want to create three shim plates at the top flange, and you want to use **Shim plate 1** twice and **Shim plate 2** once, enter 1 1 2. The first number you enter is the shim plate closest to the brace flange.

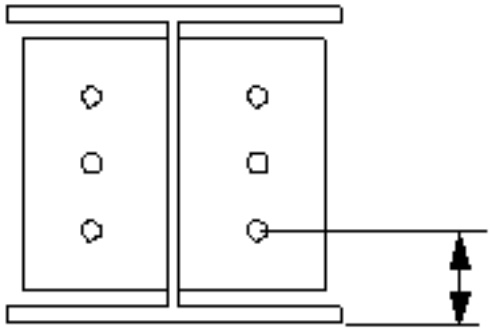
Brace bolts tab

Use the **Brace bolts** tab to control the bolts that connect the braces to the gusset plate.

Bolt group dimensions



	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Dimension for vertical bolt group position.
5	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part. 

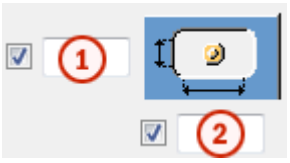
	Description
	• **Below**: From the lower edge of the secondary part to the lowest bolt. 
6	Select the bolt type.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

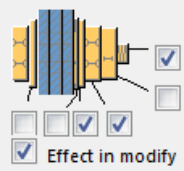
Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

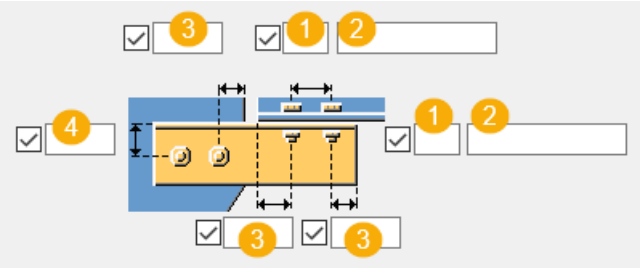
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Angle bolts tab

Use the **Angle bolts** tab to control the bolts that connect the clip angles.

Bolt group dimensions



	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Dimension for vertical bolt group position.

Vertical bolt position

Option	Description
	Bolt position from the L profile edge.
	Bolt position from the secondary part center line.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Create welds

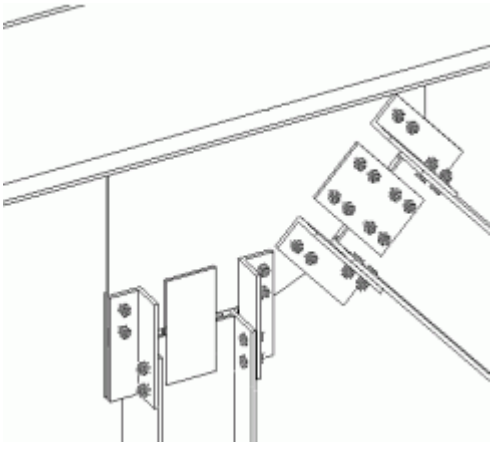
Gusseted cross (62)

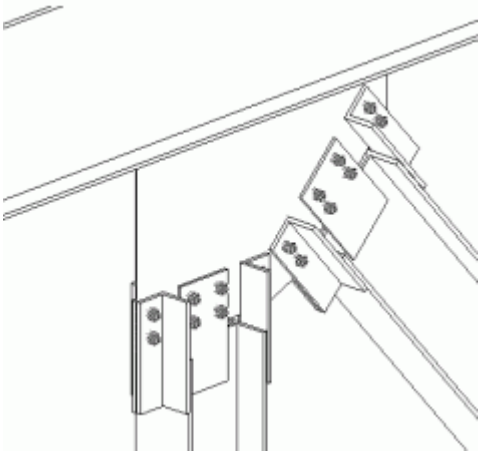
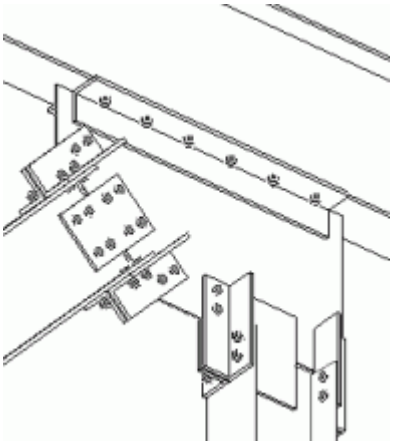
Gusseted cross bolts 1 to 10 braces to a beam or column using a gusset plate. The gusset plate is welded or bolted to the beam or column. Braces are bolted to the gusset plate using clip angles and connection plates.

Objects created

- Gusset plate
- Clip angles
- Connection plates
- Filler plates
- Shim plates
- Stiffeners
- Bolts
- Cuts
- Welds

Use for

Situation	Description
	Brace profile: W Gusset plate is welded to the beam. Different connection materials are used for each brace.

Situation	Description
	Brace profile: W Gusset plate is welded to the beam. Braces are bolted to the gusset plate using connection plates and clip angles.
	Brace profile: W Gusset plate is welded to the beam. Different connection materials are used for each brace.

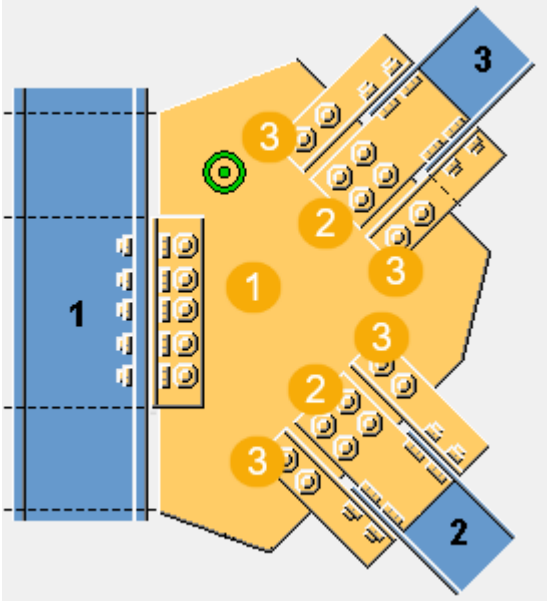
Before you start

Create a beam or column, and 1 to 10 braces.

Selection order

1. Select the main part (beam or column).
2. Select the secondary part (first brace).
3. Select the second secondary part (second brace).
4. Select the subsequent secondary parts (subsequent braces).
5. Click the middle mouse button to create the component.

Part identification key



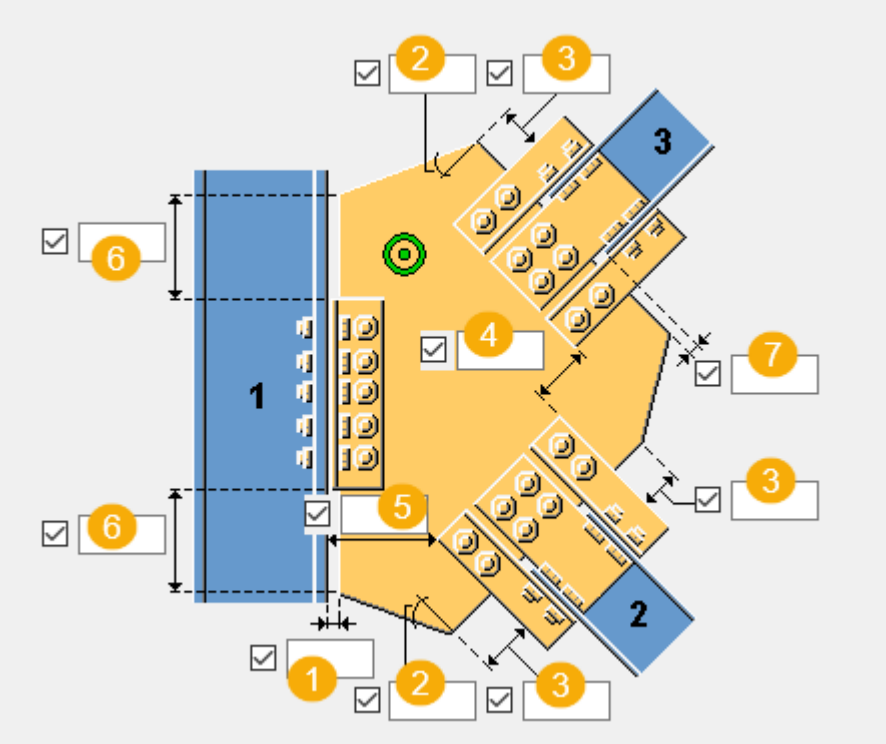
	Description
1	Gusset plate
2	Connection plate
3	Clip angle

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab

Use the **Picture** tab to define the dimensions that control the position and shape of the gusset plate.

Dimensions



	Description
1	Define the gap distance between the gusset plate edge and the main part.
2	Define the gusset plate corner angle (in degrees). This value affects the gusset plate shape.
3	Define the length of the edge of the gusset plate. This value affects the gusset plate shape.
4	Define the distance between the braces.
5	Define the distance between the main part and the brace.
6	Define the distance between the clip angle or connection plate edge and the gusset plate edge.
7	Define the gap distance between the gusset plate edge and the brace.

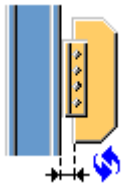
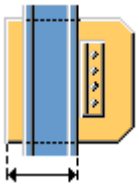
NOTE The following examples show only some of the available options. You will find more options on the **Picture** tab.

Gusset plate positioning

Define how the gusset plate is positioned when a base plate is used.

Option	Description
	Default Gusset plate is parallel to the main part. AutoDefaults can change this option.
	Gusset plate is parallel to the brace.
	Gusset plate is parallel to the brace, without a base plate.

Gusset plate dimensions

Option	Description
	Default Gusset plate does not run through the main part. Define the gusset plate cut dimension. AutoDefaults can change this option.
	Gusset plate does not run through the main part. Define the gusset plate cut dimension.
	Gusset plate runs through the main part. Define the gusset plate extension dimension.

Gusset tab

Use the **Gusset** tab to control the gusset plate properties, shape and position, connection plate, and clip angle properties and orientation.

Plates

Option	Description
Gusset	Thickness, width and height of the gusset plate.
Connection plates	Thickness and width of the connection plate.
L profile	Select the clip angle profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

NOTE The following examples show only some of the available options. You will find more options on the **Gusset** tab.

Gusset plate connection

Define how the gusset plate is connected to the main part.

Option	Description
	Default Gusset plate is welded directly to the main part. AutoDefaults can change this option.
	Gusset plate is connected to the main part with clip angles. Select to which side of the gusset plate the clip angles are created.
	Gusset plate is connected to the main part with a connection plate. Select to which side of the gusset plate the connection plate is created.

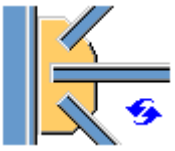
Clip angle orientation

Define how the clip angle is placed on the connection.

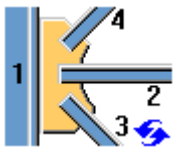
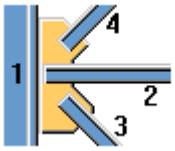
Option	Description
	Default Clip angle is placed on the connection so that the longer leg is connected to the gusset plate. AutoDefaults can change this option.
	Clip angle is placed on the connection so that the longer leg is connected to the main part.

Gusset plate shape

When you select the option to optimize the gusset plate weight, you can define whether the selection order of the braces affects the position of the braces.

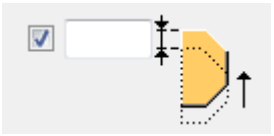
Option	Description
	Default AutoDefaults can change this option.
	This option optimizes the gusset plate weight.

Brace position

Option	Description
	Default The brace position is not affected. AutoDefaults can change this option.
	The first selected brace is placed closest to the main part.

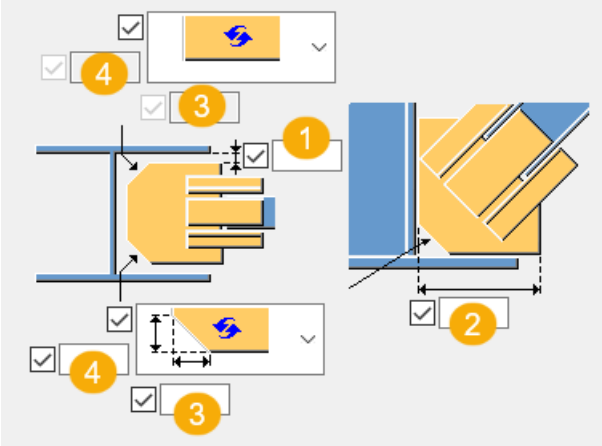
Gusset plate position on the brace

Define where to place the gusset plate on the brace. If needed, you can fine-tune the gusset plate position by moving the gusset plate in the z or in the y direction.

Option	Description
	Default Gusset plate is positioned in the middle of the brace. AutoDefaults can change this option.
	Gusset plate is positioned on the top flange of the brace.
	Define how much the gusset plate is moved in the z direction.
	Define how much the gusset plate is moved in the y direction.

Gusset plate chamfer

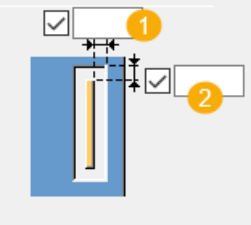
Define the gusset plate chamfer type and dimensions, and the angle limit for parallel bracing.



1	Distance between the connection plate and the inner flange of the main part.
2	Horizontal distance between the gusset plate edge and the flange of the main part.
3	Horizontal dimension of the chamfer. By default, the second chamfer is not created.
4	Vertical dimension of the chamfer. By default, the second chamfer is not created.

Cut size

If the gusset plate runs through the main part, define the size of the cut created for the gusset plate.



	Description
1	Define the horizontal size of the cut.
2	Define the vertical size of the cut.

Brace conn tab

Use the **Brace conn** tab to define the connection plate, clip angle, and filler plate properties. Select whether twin profiles are used for the angle connection.

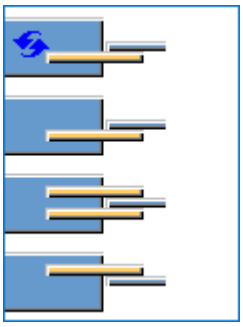
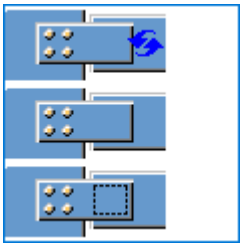
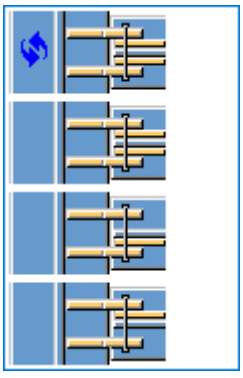
Parts

Part	Description
Connect plate	Thickness and width of the connection plate.
Connection profile	Select the profile from the profile catalog.
L prof1 to gus. pl., **L prof2 to gus.pl.**	Select the clip angle profile from the profile catalog.
Filler plate	Thickness and height of the filler plate.
Plate1 to gus.pl	Thickness, width and height of the plate.
Plate2 to gus.pl	Height of the plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

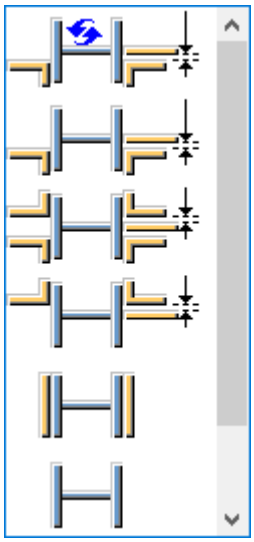
Option	Description	Default
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

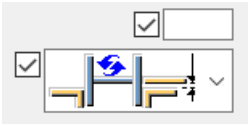
Plate creation

Option	Description
	Select whether one or two connection plates are created.
	Select whether a filler plate is created between the connection plate and the brace web. The default is that a filler plate is not created.
	Select the filler plate creation side. You can use this option when you have selected to create two connection plates.

Clip angle creation

Define whether the braces are attached to the gusset plate using clip angles or shear tabs, and specify the number of clip angles to create. The default option is to create two clip angles below the brace web.



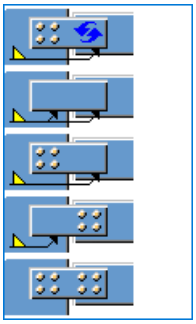


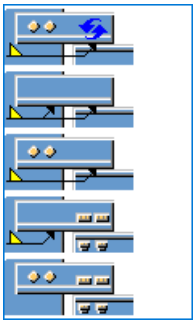
Clip angle orientation

Define how the clip angle is placed on the connection.

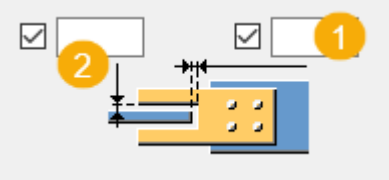
Option	Description
	Default Clip angle is placed on the connection so that the longer leg is connected to the gusset plate. AutoDefaults can change this option.
	Clip angle is placed on the connection so that the longer leg is connected to the main part.

Connection type

Option	Description
	Select the connection type (weld or bolts) between the gusset plate and the connection plate.

Option	Description
	Select the connection type (weld or bolt) between the gusset plate and the L profile.

Connection plate gap dimensions



	Description
1	Horizontal gap dimension
2	Vertical gap dimension

Shim plates tab

Use the **Shim plates** tab to define shim plate properties.

Plates

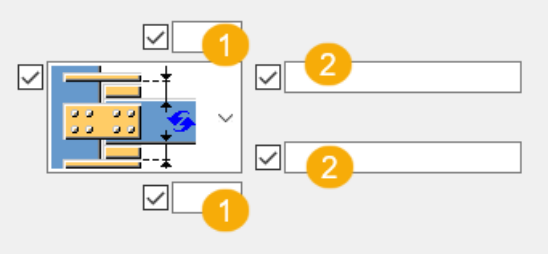
Option	Description
Shim plate 1, **Shim plate 2**, **Shim plate 3**	Thickness, width and height of the shim plates.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in

Option	Description	Default
		File menu --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Shim plate position and number

You can create shim plates when connecting braces to the gusset plate using clip angles.

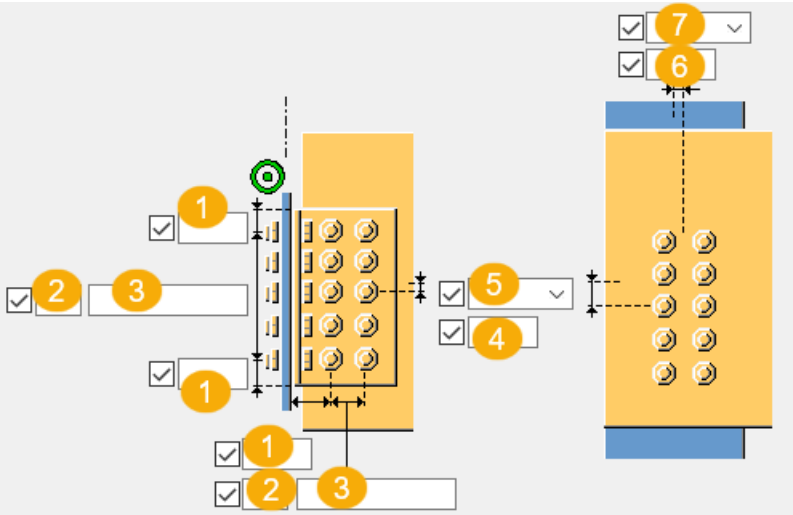


	Description
1	Define the gap between the brace and connection plate.
2	Define how many shim plates are created at the top and bottom flanges. Enter the shim plate profile numbers: 1, 2 or 3. These are the numbers that are on the upper part on the **Shim plates** tab. For example, if you want to create three shim plates at the top flange, and you want to use **Shim plate 1** twice and **Shim plate 2** once, enter 1 1 2. The first number you enter is the shim plate closest to the brace flange.

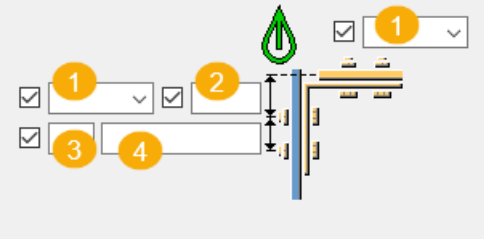
Gusset conn tab

Use the **Gusset conn** tab to control the bolt group properties for bolts that connect the gusset plate to the main part, and to control the clip angle attachment.

Bolt group dimensions



	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Dimension for vertical bolt group position.
5	Select how to measure the dimensions for vertical bolt group position.
6	Dimension for horizontal bolt group position.
7	Select how to measure the dimensions for horizontal bolt group position.

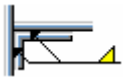


	Description
1	Location where the bolts should be attached.

	Description
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Clip angle attachment type

Define how the clip angle is attached to the gusset plate and to the main part.

Option	Description
	Default Main part is bolted and secondary part is welded. AutoDefaults can change this option.
	Automatic When the main part is a tube profile, the clip angles are welded to the main part and bolted to the secondary part. Otherwise the clip angles are bolted to both parts.
	Main part is bolted and secondary part is welded.
	Main part is welded and secondary is part bolted.
	Both parts are bolted.
	Both parts are welded.

Bolts on gusset plate

Define whether the gusset plate is connected to the main part with bolts when no clip angles are used.

	Description
	Default Bolts are not created in the gusset plate. AutoDefaults can change this option.

	Description
	Bolts are created in the gusset plate.

Staggering of bolts

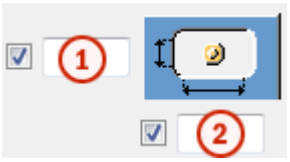
Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.

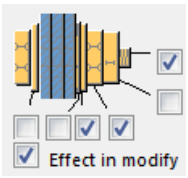


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

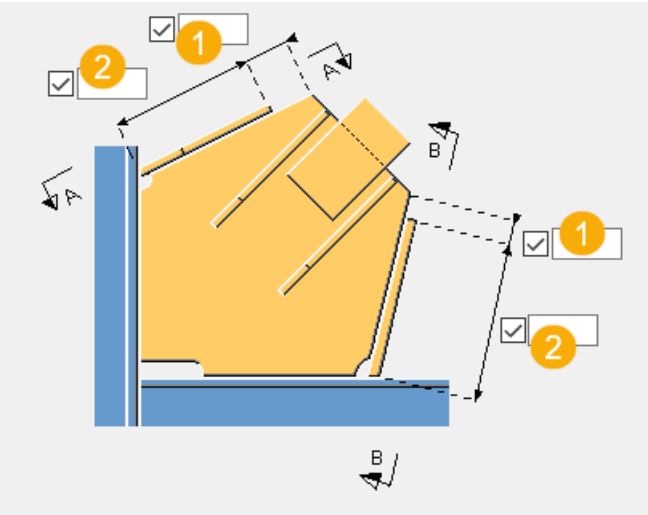
Stiffeners tab
Use the **Stiffeners** tab to define the stiffener properties and dimensions.

Parts

Part	Description
Stiffener 1, **Stiffener 2**	Thickness of the stiffener.

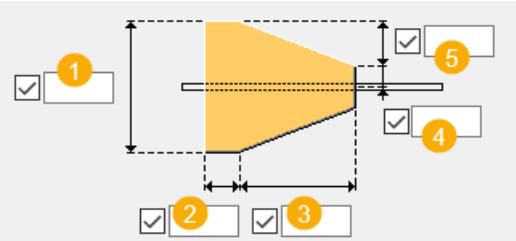
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Stiffener length



1	Define the distance between the stiffener edge and the gusset plate edge.
2	Define the length of stiffener.

Stiffener dimensions

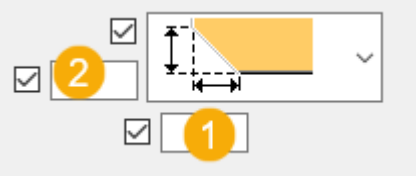


	Description
1	Define the width of the stiffener.
2	Define the length of the stiffener base.
3	Define the length of the skew part of the stiffener.
4	Define the distance from the stiffener center line.
5	Define the vertical distance between the stiffener base and the skew part.

Chamfer type

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

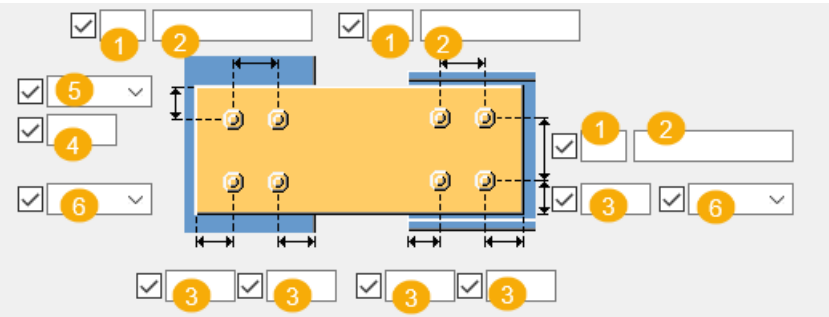
Chamfer dimensions

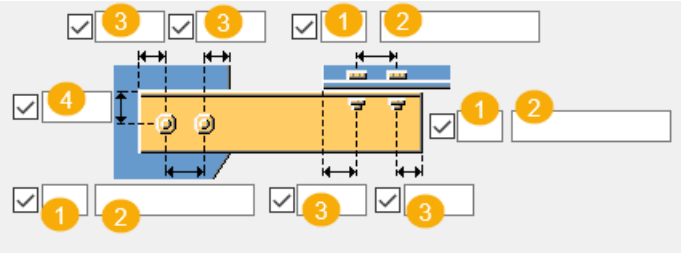


	Description
1	Horizontal dimension of the chamfer.
2	Vertical dimension of the chamfer.

Brace bolts 1 / Brace bolts 2 / Brace bolts 3 tab
Use the **Brace bolts 1**, **Brace bolts 2** and **Brace bolts 3** tabs to control the bolts that connect the first, the second, and the subsequent braces to the gusset plate.

Bolt group dimensions





	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.

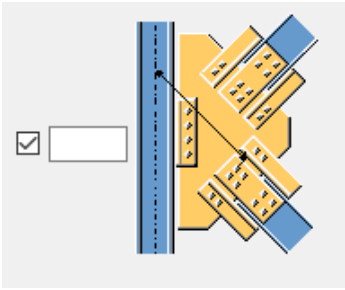
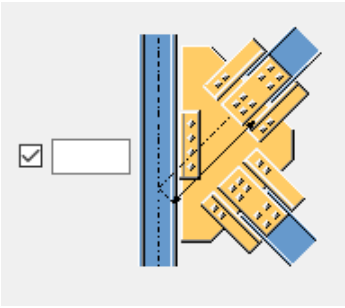
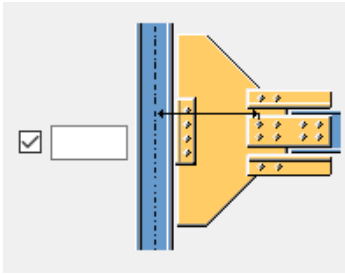
	Description
4	Dimension for vertical bolt group position.
5	Select how to measure the dimensions for vertical bolt group position.
6	Select the bolt type.

Vertical bolt position

Option	Description
	Bolt position from the L profile edge.
	Bolt position from the secondary part center line.

Bolt distance

Define the minimum distance from the connection plate bolts to the intersection point of the main part and brace center lines. If a brace is perpendicular to the main part, the distance is measured from the main part center line to the nearest bolts.

Option	Option	Option
		

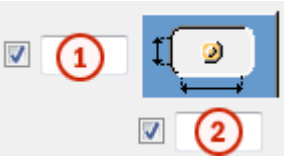
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the	`Yes`

Option	Description	Default
	bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered

Option	Description
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

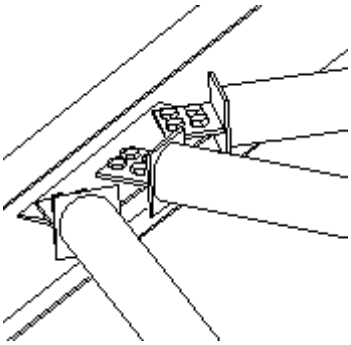
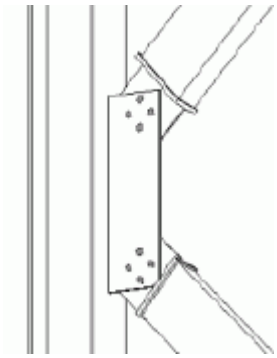
Portal bracing (105)

Portal bracing (105) connects up to 3 hollow braces to a beam or column using a gusset plate and tees.

Objects created

- Gusset plate
- Tees
- Bolts
- Welds

Use for

Situation	Description
	Brace profile: Tube Gusset plate is welded to the beam. Braces are bolted to the gusset plate using tees.
	Brace profile: RHS Gusset plate is welded to the column. Braces are bolted to the gusset plate using tees.

Limitations

The braces must be in the same plane.

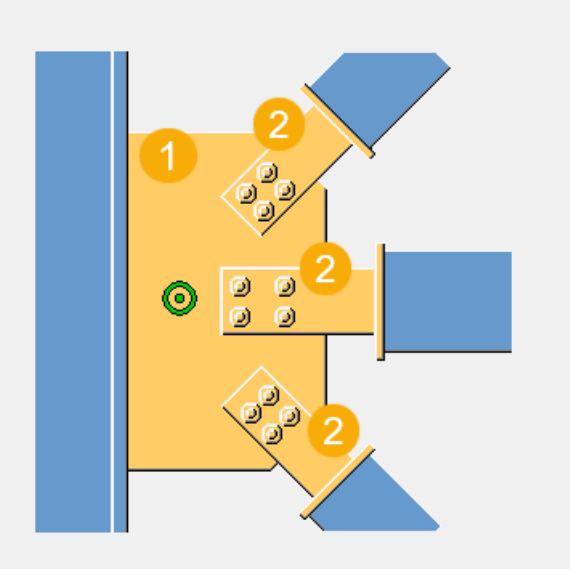
Before you start

Create a beam, and up to three braces all in the same plane.

Selection order

1. Select the main part (beam or column).
2. Select the secondary part (first brace).
3. Select the second secondary part (second brace).
4. Select the subsequent secondary part (third brace).
5. Click the middle mouse button to create the component.

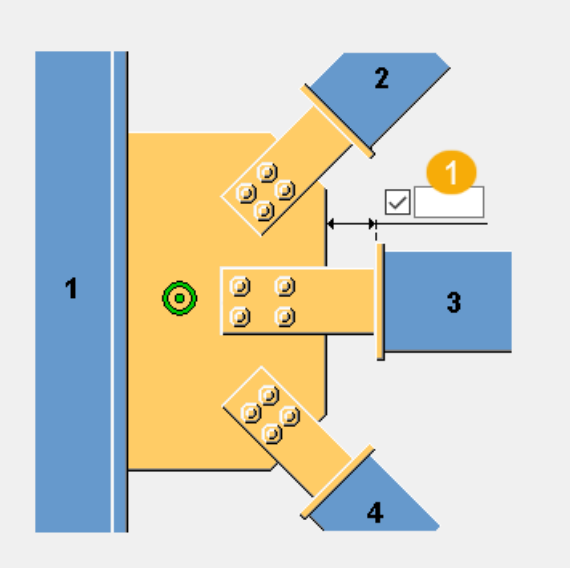
Part identification key



	Description
1	Gusset plate
2	Tee

Picture tab

Use the **Picture** tab to define the tee dimension from the gusset plate edge.



	Description
1	Define the tee edge dimension.

Parts tab
Use the **Parts** tab to define the properties of the gusset plate, and the tees at the ends of the braces. You can define the tees separately for each brace.

Parts

Option	Description
Gusset	Thickness, width and height of the gusset plate.
Tee flange	To have Tekla Structures use two plates to create the tee, leave the tee profile on the **Parts** tab empty. You must enter the thickness, width, and height dimensions for the tee flange and web instead. Define the thickness, width and height of the tee flange.
Tee web	To have Tekla Structures use two plates to create the tee, leave the tee profile on the **Parts** tab empty. You must enter the thickness, width, and height dimensions for the tee flange and web instead. Define the thickness, width and height of the tee web.
Tee profile	Select the tee profile from the profile catalog. You can use a T profile or an I profile. If you use an I profile, the component creates the tee by cutting the I profile. You must specify where to cut the I profile, using the tee depth option on the **Parameters** tab.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

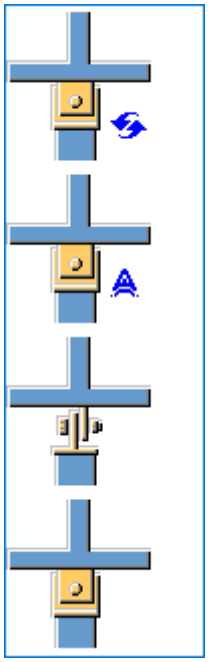
Option	Description	Default
Class	Part class number.	

Parameters tab
Use the **Parameters** tab to define the tee dimensions, clearances, gusset plate position and chamfers.

Option	Description
Min. cut angle (0-90 deg)	Define the minimum cut angle.
Define welds for each Tee	Select whether to use welds for each tee.
Define bolts for each Tee	Select whether to use bolts for each tee.

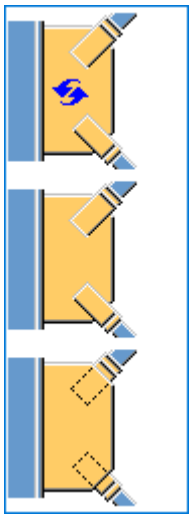
Gusset position

Select the gusset position on the brace.

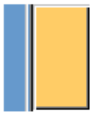


Tee position

Select the tee position on the gusset.

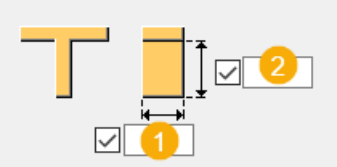


Gusset chamfer

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Chamfers are created.

Tee dimensions

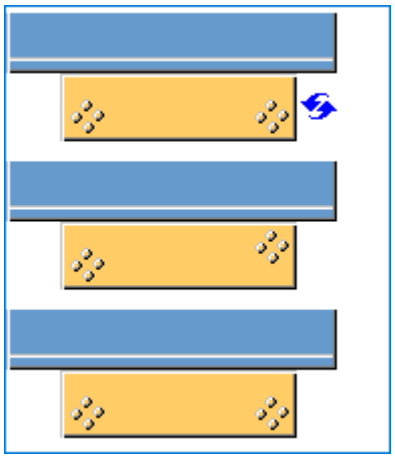
Define the tee dimensions for each brace.



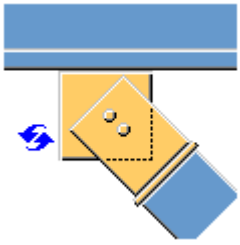
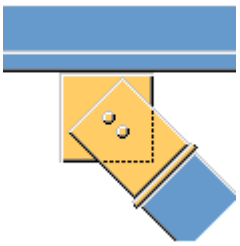
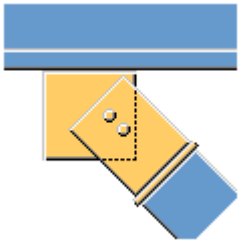
	Description
1	Tee length
2	Tee depth

Bolt group alignment

Select whether bolt groups are aligned.

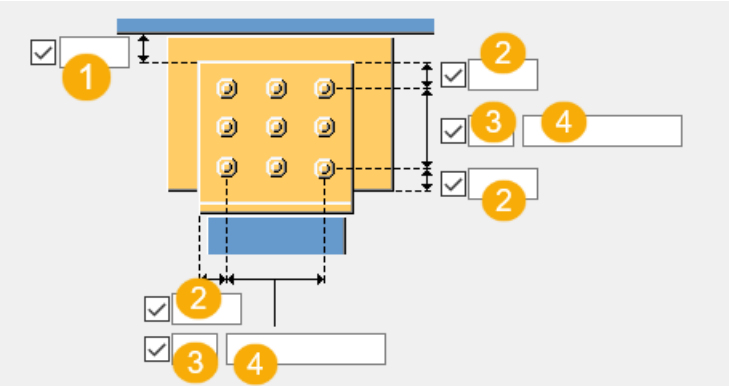


Gusset position on the main part

Option	Description
	Default Gusset is positioned according to the defined distances. AutoDefaults can change this option.
	Gusset is positioned according to the defined distances.
	Gusset is centered to the center of the bolt group.
	Gusset is centered to the center of the connection origin.

Bolts 1 / Bolts 2 / Bolts 3 tabs
Use the **Bolts 1**, **Bolts 2** and **Bolts 3** tabs to control the bolts that connect the first, the second, and the third brace to the gusset plate.

Bolt group dimensions



	Description
1	Plate edge distance to the main part web.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

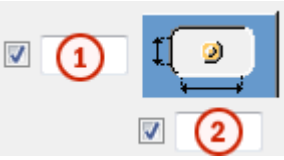
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when	`Yes`

Option	Description	Default
	bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

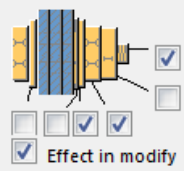


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

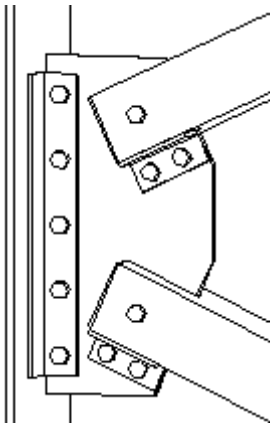
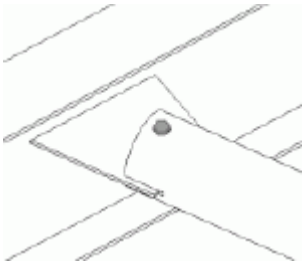
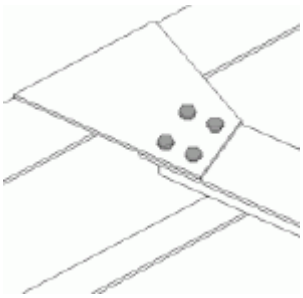
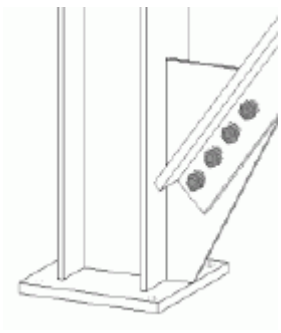
Bolted gusset (196)

Bolted gusset (196) connects 1 to 10 braces to a beam or column using a gusset plate, which is bolted or welded to the beam or column. The braces are bolted to the gusset plate. Clip angles can be created either at the ends of the braces, or on each side.

Objects created

- Gusset plate
- Clip angles or shear tabs (optional) that connect the gusset plate to the beam or column
- Clip angles (optional) that connect the brace to the gusset plate
- Connection plates
- Seal plates (hollow braces)
- Stiffeners (optional)
- Bolts
- Welds
- Cuts

Use for

Situation	Description
	Brace profile: RHS Gusset plate is bolted to the beam flange using a clip angle. Braces are slotted around the gusset plate and attached to it using bolts and clip angles.
	Brace profile: Tube Gusset plate is welded to the beam web. Brace is notched around the gusset plate and pinned to it.
	Brace profile: T Gusset plate is welded to the beam flange. Brace is bolted to the gusset plate.
	Brace profile: L Gusset plate is welded to the column flange. Brace is bolted to the gusset plate.

Before you start

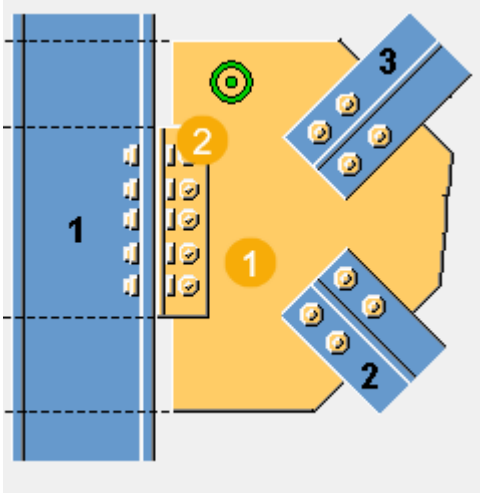
Create a beam or column, and 1 to 10 braces.

Selection order

1. Select the main part (beam or column).

2. Select the secondary part (first brace).
3. Select the second secondary part (second brace).
4. Select the subsequent secondary parts (subsequent braces).
5. Click the middle mouse button to create the component.

Part identification key

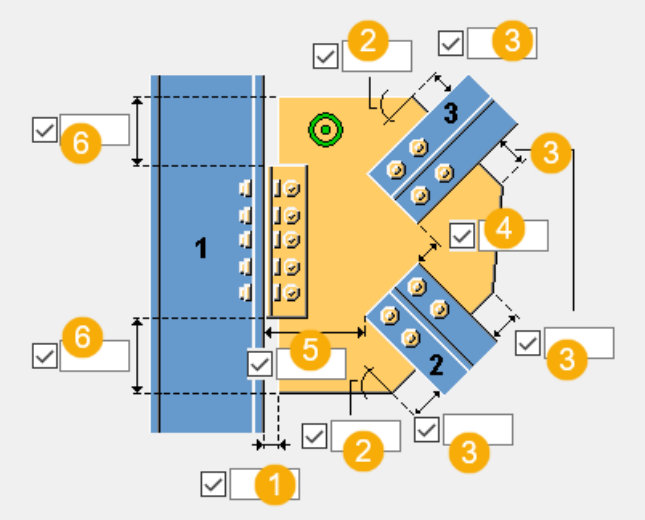


1	Gusset plate
2	Clip angle

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Picture tab
Use the **Picture** tab to define the dimensions that control the position and shape of the gusset plate.

Dimensions



	Description
1	Define the gap distance between the gusset plate edge and the main part.
2	Define the gusset plate corner angle (in degrees). This value affects the gusset plate shape.
3	Define the length of the edge of the gusset plate. This value affects the gusset plate shape.
4	Define the distance between the braces.
5	Define the distance between the main part and the brace.
6	Define the distance between the clip angle or connection plate edge and the gusset plate edge.

NOTE The following examples show only some of the available options. You will find more options on the **Picture** tab.

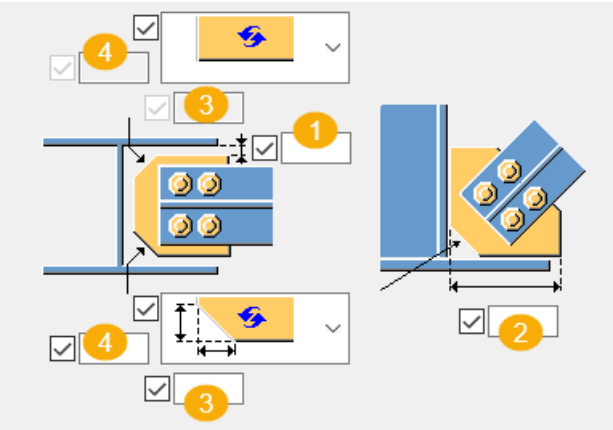
Gusset plate positioning

Define how the gusset plate is positioned when a base plate is used.

Option	Desription
	Default Gusset plate is parallel to the main part. AutoDefaults can change this option.
	Gusset plate is parallel to the brace.

Gusset plate chamfer

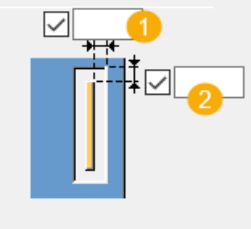
Define the gusset plate chamfer type and dimensions, and the angle limit for parallel bracing.



	Description
1	Distance between the connection plate and the inner flange of the main part.
2	Horizontal distance between the gusset plate edge and the flange of the main part.
3	Horizontal dimension of the chamfer. By default, the second chamfer is not created.
4	Vertical dimension of the chamfer. By default, the second chamfer is not created.

Cut size

If the gusset plate runs through the main part, define the size of the cut created for the gusset plate.



	Description
1	Define the horizontal size of the cut.
2	Define the vertical size of the cut.

Gusset tab
Use the **Gusset** tab to control the gusset plate properties, shape and position, clip angle properties and orientation, and angle limit for parallel bracing.

Plates

Option	Description
Gusset	Thickness, width and height of the gusset plate.
Connection plates	Thickness and width of the connection plate.
L profile	Select the clip angle profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

NOTE The following examples show only some of the available options. You will find more options on the **Gusset** tab.

Gusset plate connection

Define how the gusset plate is connected to the main part.

Option	Description
	Default Gusset plate is welded directly to the main part. AutoDefaults can change this option.

Option	Description
	Gusset plate is connected to the main part with clip angles. Select to which side of the gusset plate the clip angles are created.
	Gusset plate is connected to the main part with a connection plate. Select to which side of the gusset plate the connection plate is created.

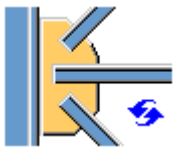
Clip angle orientation

Define how the clip angle is placed on the connection.

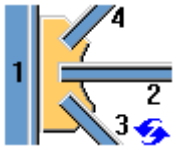
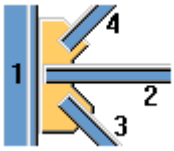
Option	Description
	Default Clip angle is placed on the connection so that the longer leg is connected to the gusset plate. AutoDefaults can change this option.
	Clip angle is placed on the connection so that the longer leg is connected to the main part.

Gusset plate shape

When you select the option to optimize the gusset plate weight, you can define whether the selection order of the braces affects the position of the braces.

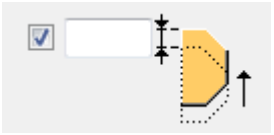
Option	Description
	Default AutoDefaults can change this option.
	This option optimizes the gusset plate weight.

Brace position

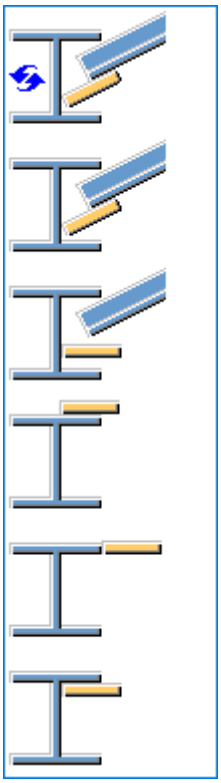
Option	Description
	Default The brace position is not affected. AutoDefaults can change this option.
	The first selected brace is placed closest to the main part.

Gusset plate position on the brace

Define where to place the gusset plate on the brace. If needed, you can fine-tune the gusset plate position by moving the gusset plate in the z or in the y direction.

Option	Description
	Default Gusset plate is positioned in the middle of the brace. AutoDefaults can change this option.
	Gusset plate is positioned on the top flange of the brace.
	Define how much the gusset plate is moved in the z direction.
	Define how much the gusset plate is moved in the y direction.

Gusset plate position on the column or beam

Option	Description
	Select the gusset position. The default option is parallel to the brace.

Brace conn tab

Use the **Brace conn** tab to define the seal plate, brace notch and slot properties.

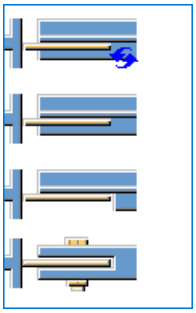
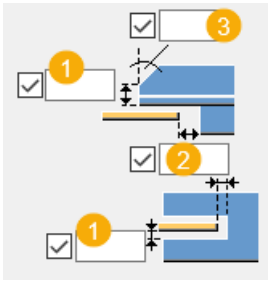
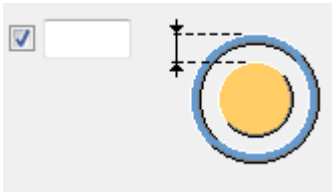
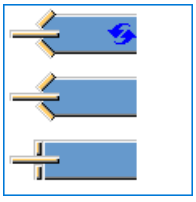
Seal plate

Option	Description
Seal plate	Thickness of the seal plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part**

Option	Description	Default
		material box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

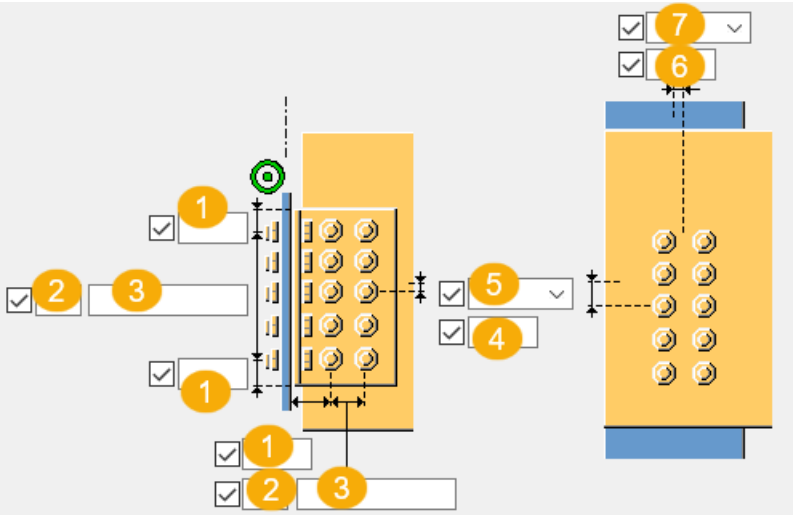
Brace notch

Option	Description
	Select whether the brace is notched.
	1. Vertical notch dimension. 2. Horizontal notch dimension. 3. Notch angle.
	Define the plate distance from the brace outer edge.
	Select whether to create bevel cuts at the brace ends or whether to create a square brace end.

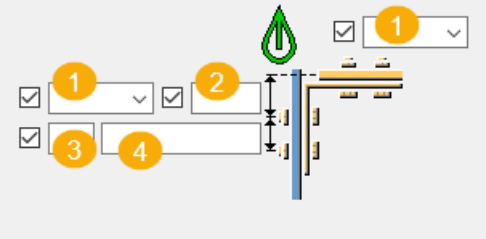
Gusset bolts tab

Use the **Gusset bolts** tab to control the bolt group properties for bolts that connect the gusset plate to the main part, and to control the clip angle attachment.

Bolt group dimensions



1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Dimension for vertical bolt group position.
5	Select how to measure the dimensions for vertical bolt group position.
6	Dimension for horizontal bolt group position.
7	Select how to measure the dimensions for horizontal bolt group position.



1	Location where the bolts should be attached.

2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Clip angle attachment type

Option	Description
	Default Both parts are bolted. AutoDefaults can change this option.
	Automatic When the main part is a tube profile, the clip angles are welded to the main part and bolted to the secondary part. Otherwise the clip angles are bolted to both parts.
	Main part is bolted and secondary part is welded.
	Main part is welded and secondary is part bolted.
	Both parts are bolted.
	Both parts are welded.

Bolts on gusset plate

Define whether the gusset plate is connected to the main part with bolts when no clip angles are used.

Option	Description
	Default Bolts are not created in the gusset plate. AutoDefaults can change this option.

Option	Description
	Bolts are created in the gusset plate.

Staggering of bolts

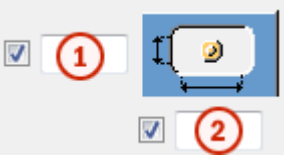
Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.

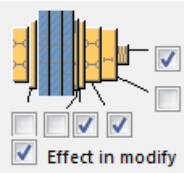


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



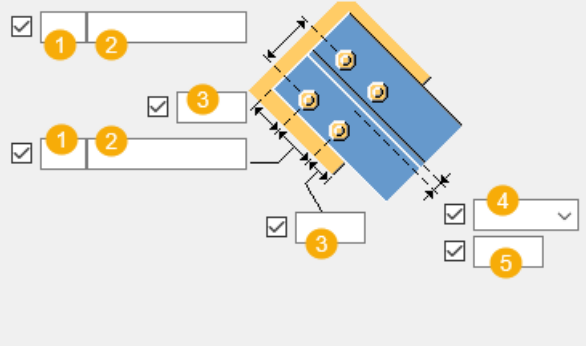
To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

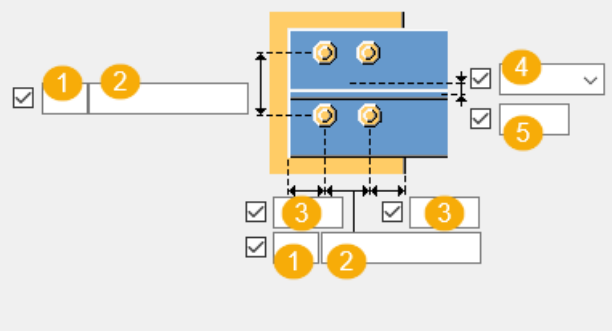
Bolt length increase

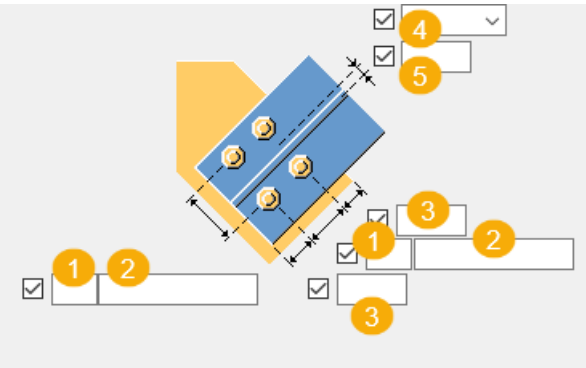
Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Brace bolts 1 / Brace bolts 2 / Brace bolts 3 tab
Use the **Brace bolts 1**, **Brace bolts 2** and **Brace bolts 3** tabs to control the bolts that connect the first, the second, and the subsequent braces to the gusset plate.

Bolt group dimensions





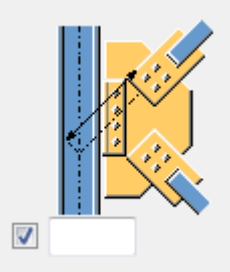


	Description
1	Number of bolts.

	Description
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Select how to measure the dimensions for vertical bolt group position.
5	Dimension for vertical bolt group position.

Bolt distance

Define the minimum distance from the connection plate bolts to the intersection point of the main part and brace center lines. If a brace is perpendicular to the main part, the distance is measured from the main part center line to the nearest bolts.



Attributes for Tube crossing (22)

With hollow secondary parts, **Bolted gusset (196)** automatically removes connection objects created between the gusset plate and the hollow secondary part, and then uses the **Tube crossing (22)** connection between the gusset plate and the hollow secondary part.

To define which property file that **Tube crossing (22)** uses, go to the **Brace bolts 2** tab, and enter the property file name in the **Attributes for Tube crossing (22)** box.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered

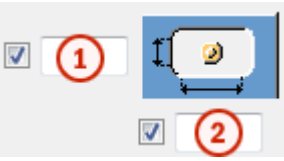
Option	Description
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.

Option	Description	Default
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Angle bolts tab

Use the **Angle bolts** tab to control the bolts that connect the clip angles.

Part

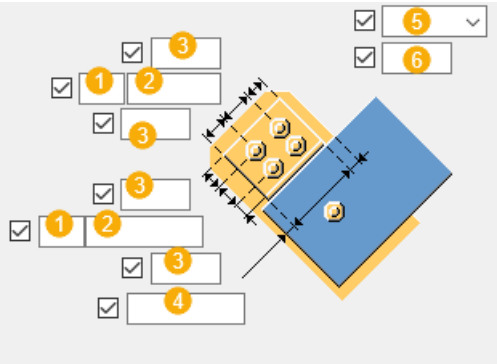
Option	Description
L profile	Select the clip angle profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

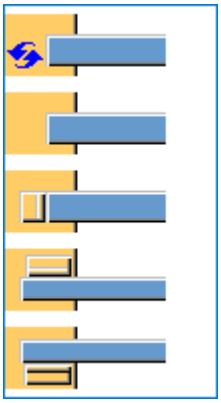
Bolt group dimensions



	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Define the edge distance between the clip angle and the brace.

	Description
5	Select how to measure the dimensions for vertical bolt group position.
6	Dimension for vertical bolt group position.

Clip angle position

Option	Description
	Select the clip angle position.

Stiffeners tab

Use the **Stiffeners** tab to define the stiffener properties and dimensions.

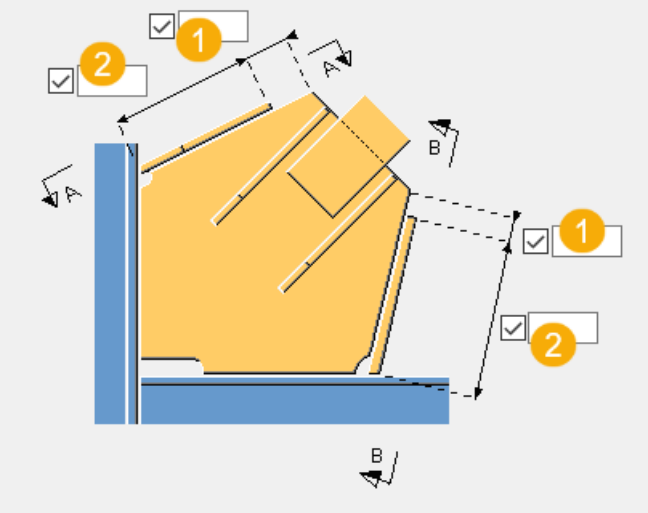
Parts

Part	Description
Stiffener 1, **Stiffener 2**	Thickness of the stiffener.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

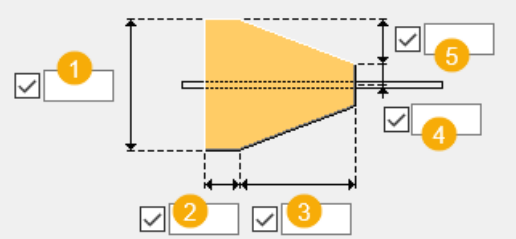
Option	Description	Default
Finish	Describes how the part surface has been treated.	

Stiffener length



	Description
1	Define the distance between the stiffener edge and the gusset plate edge.
2	Define the length of stiffener.

Stiffener dimensions

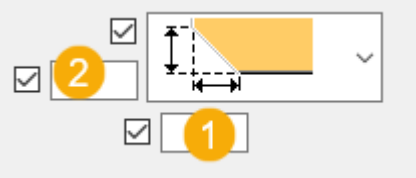


	Description
1	Define the width of the stiffener.
2	Define the length of the stiffener base.
3	Define the length of the skew part of the stiffener.
4	Define the distance from the stiffener center line.
5	Define the vertical distance between the stiffener base and the skew part.

Chamfer type

Option	Description
	Default No chamfer AutoDefaults can change this option.
	No chamfer
	Line chamfer
	Convex arc chamfer
	Concave arc chamfer

Chamfer dimensions



	Description
1	Horizontal dimension of the chamfer.
2	Vertical dimension of the chamfer.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

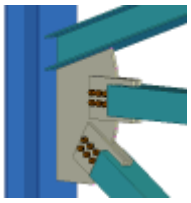
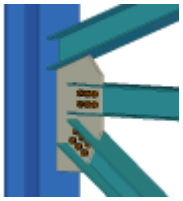
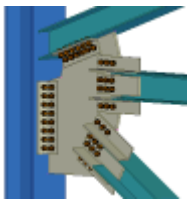
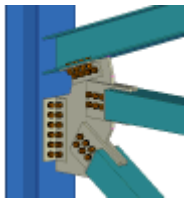
Create welds

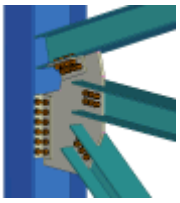
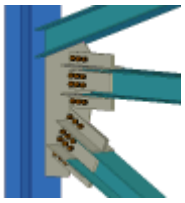
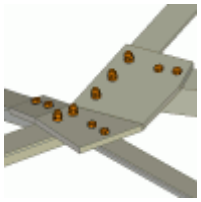
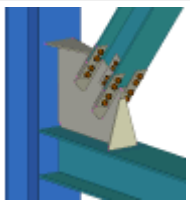
Corner bracing connections

Use corner bracing connections to automatically connect bracing where two parts meet to form a corner, for example:

- Beam and column
- Column and base plate
- Beam and extended end plate
- Two beams and a column

Tekla Structures includes the following corner bracing connections:

Component	Image	Description
Corner tube gusset (56) (page 3023)		Bolts braces to a gusset plate using connection plates and optional tongue plates. Seals hollow braces.
Corner bolted gusset (57) (page 3039)		Bolts braces to a gusset plate. Welds the gusset plate to one or both of the parts that form the corner. Seals hollow braces.
Wraparound gusset (58) (page 3054)		Bolts or welds braces to a gusset plate. Option to wrap the gusset plate around a third part, usually a column. Option to use connection material to connect the gusset plate to the main parts.
Hollow brace wraparound gusset (59) (page 3081)		Bolts or welds hollow braces to a gusset plate. Option to wrap the gusset plate around a third part, usually a column. Option to use connection material to

Component	Image	Description
		connect the gusset plate to the main parts.
Wraparound gusset cross (60) (page 3102)		Bolts or welds braces to a gusset plate. Option to use clip angles. Option to wrap the gusset plate around a third part, usually a column. Option to use connection material to connect the gusset plate to the main parts.
Corner wrapped gusset (63) (page 3123)		Bolts braces to a gusset plate using clip angles and connection plates. Welds the gusset plate to one of the main parts.
Bent gusset (140) (page 3142)		Connects braces in different planes to one or more beams or columns that are in different planes.Creates a gusset plate that is bent along two different bending lines. Seals hollow braces.
Heavy brace (165) (page 3144)		Bolts a single brace to a gusset plate where two parts meet to form a corner. Rib plates strengthen the connection.

Corner tube gusset (56)

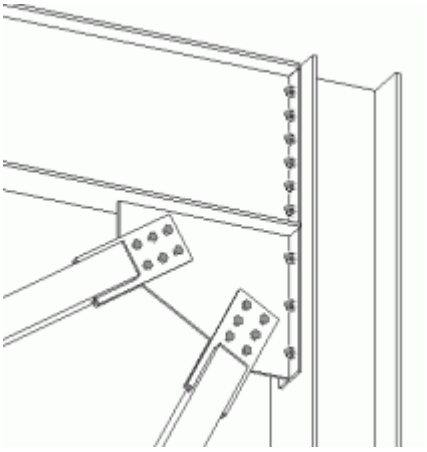
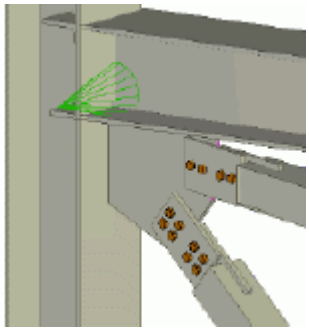
Corner tube gusset (56) connects 1 to 10 hollow braces to the corner where two parts meet, using a gusset plate. Bolts braces to the gusset plate using a connection plate or tongue plate slotted into the brace. Option to create extra connection plates. Seals braces.

Objects created

- Gusset plate
- Connection plates
- Seal plates
- Tongue plates (optional)
- Cover plates (optional)

- Stiffeners
- Bolts
- Welds

Use for

Situation	Description
	Brace profile: RHS Framing type: Column and extended end plate Gusset plate is welded to an extended end plate. Braces are bolted to the gusset plate using tongue plates.
	Brace profile: RHS Framing type: Column and beam Gusset plate is welded to the column flange. Braces are bolted to the gusset plate using tongue plates, and connection and cover plates.

Before you start

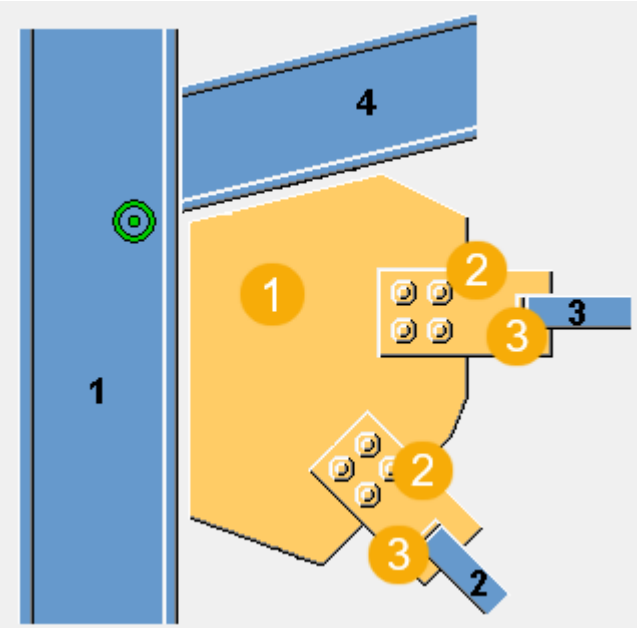
Create two parts that form a corner, and 1 to 10 braces.

Selection order

1. Select the main part (the first part that forms the corner).
2. Select the secondary part (first brace).
3. Select the second secondary part (second brace).
4. Select the subsequent secondary parts (subsequent braces).
5. Select the secondary part that forms the corner (Tekla Structures connects the gusset plate to this part).
6. Click the middle mouse button to create the connection.

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Part identification key

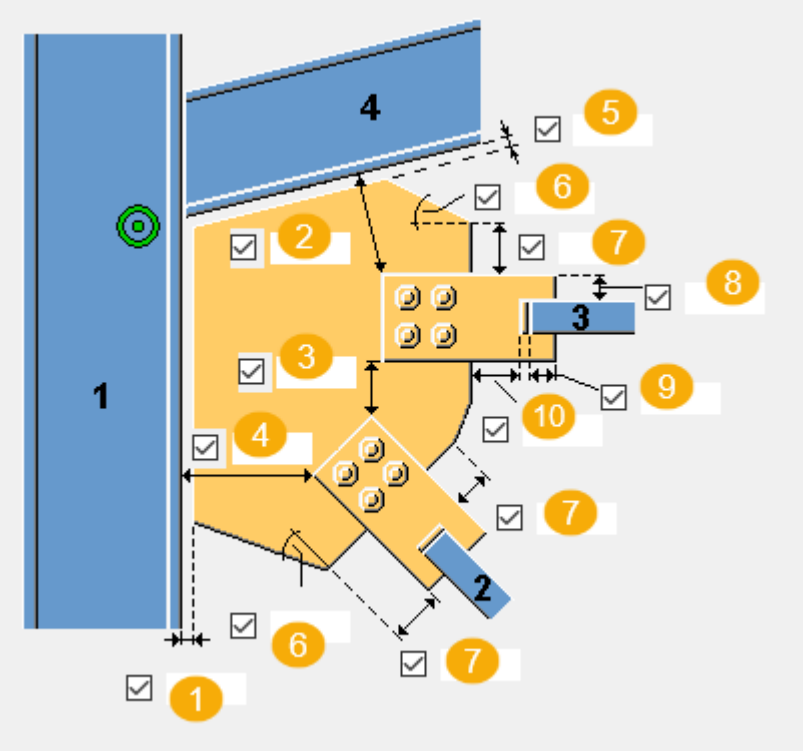


	Description
1	Gusset plate
2	Connection plate
3	Seal plate

Picture tab

Use the **Picture** tab to define the gusset plate position and shape.

Dimensions



	Description
1	Define the gap distance between the gusset plate edge and the main part (first part that forms the corner) .
2	Define the edge distance between the connection plate and the last secondary part.
3	Define the distance between the connection plates.
4	Define the edge distance between the connection plate and the main part.
5	Define the gap distance between the gusset plate edge and the secondary part (second part that forms the corner).
6	Define the gusset plate corner angle (in degrees). This value affects the gusset plate shape.
7	Define the length of the edge of the gusset plate. This value affects the gusset plate shape.
8	Define the length of the edge of the connection plate.
9	Define the brace length on the connection plate.
10	Define the edge distance between the seal plate and gusset plate.

Gusset tab
Use the **Gusset** tab to control the gusset plate properties.

Gusset plate

Option	Description
Gusset	Thickness, width and height of the gusset plate.

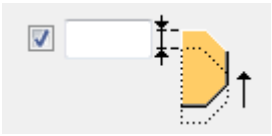
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

NOTE The following examples show only some of the available options. You will find more options on the **Gusset** tab.

Gusset plate position on the brace

Define where to place the gusset plate on the brace. If needed, you can fine-tune the gusset plate position by moving the gusset plate in the z or in the y direction.

Option	Description
	Default Gusset plate is positioned in the middle of the brace. AutoDefaults can change this option.

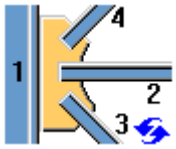
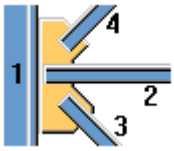
Option	Description
	Gusset plate is positioned on the top flange of the brace.
	Define how much the gusset plate is moved in the z direction.
	Define how much the gusset plate is moved in the y direction.

Gusset plate shape

When you select the option to optimize the gusset plate weight, you can define whether the selection order of the braces affects the position of the braces.

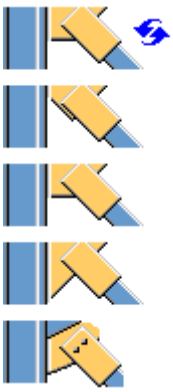
Option	Description
	Default AutoDefaults can change this option.
	This option optimizes the gusset plate weight.

Brace position

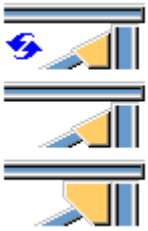
Option	Description
	Default The brace position is not affected. AutoDefaults can change this option.
	The first selected brace is placed closest to the main part.

Gusset plate shape

The gusset plate edge can be perpendicular either to the main part or the secondary part.

Option	Description
	Select the gusset plate edge shape between the last and second last secondary part.
	Select the gusset plate egde shape between the main part and the first secondary part.

Gusset plate fitting

Option	Description
	Select whether the gusset is fitted to the last selected secondary part.

Gusset welding

Weld 1 is used to weld the gusset part to the main part and weld 4 is used for welding the gusset to the last secondary part.

Option	Description
	Default Gusset plate is welded to the main part. AutoDefaults can change this option.
	Gusset plate is welded to the main part.

Option	Description
	Gusset plate is welded to the secondary part.
	Gusset plate is welded to the main part and the secondary part.

Brace conn tab

Use the **Brace conn** tab to control connection plate, seal plate, tongue plate, and cover plate properties.

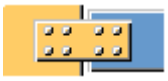
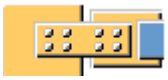
Plates

Option	Description
Connection plate	Thickness, width and height of the connection plate.
Seal plates	Thickness, width and height of the seal plate.
Tongue plate	Thickness of the tongue plate.
Cover plate	Thickness, width and height of the cover plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Brace connection types

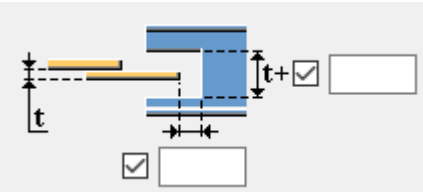
Define how the brace is connected to the connection plate.

Option	Description
	Default Brace is welded. AutoDefaults can change this option.
	Brace is welded.
	Brace is bolted.
	Brace is welded and notched around the nuts.
	Tongue plate and cover plate are created.

Cut in bracing

Define the width of the cut in the bracing, where **t** is the thickness of the connection plate.

Define the length of the cut in the bracing from the edge of the connection plate.



Round cut in bracing

Option	Description
	Default Square cut AutoDefaults can change this option.
	Square cut
	Round cut Enter the radius value.

Connection plate

Define whether the brace is notched or the connection plate cut when the connection plate is connected to the brace.

Option	Description
	Default Brace is notched. AutoDefaults can change this option.
	Brace is notched.
	Connection plate is cut.
	Connection plate is cut, but the part of the connection plate created inside the bracing is not deleted.
	If you cut the connection plate, you can define the size of the gap between the brace and the connection plate.

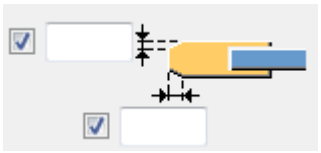
Number of connection plates

Define whether one or two connection plates are used for connecting the brace to the gusset plate.

Option	Description
	Default One connection plate AutoDefaults can change this option.
	One connection plate
	Two connection plates

Connection plate chamfer

Option	Description
	Default No chamfers are created. AutoDefaults can change this option.

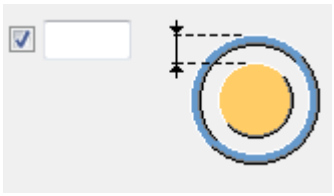
Option	Description
	No chamfers are created.
	Chamfers are created.
	If you create chamfers, define the vertical and horizontal chamfer dimensions.

End plates

If you use the end plates to seal the braces, define the end plate shape and dimensions.

Option	Description
	Default Square end plate. AutoDefaults can change this option.
	Square end plate.
	Round end plate.

End plate edge distance

Option	Description
	End plate edge distance from the brace outer edge.

Stiffeners tab

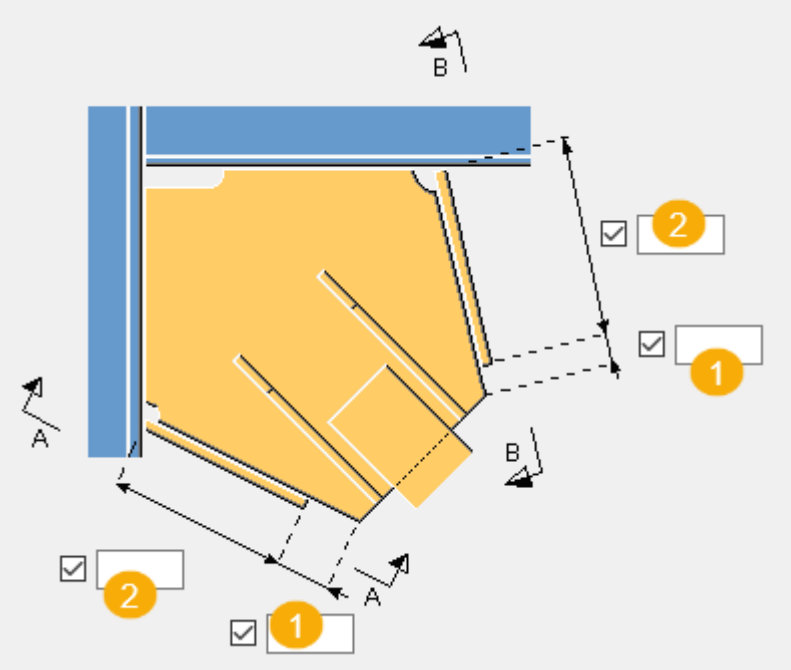
Use the **Stiffeners** tab to control the stiffener properties and dimensions.

Stiffeners

Option	Description
Stiffener 1 **Stiffener 2**	Thickness of the stiffener.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

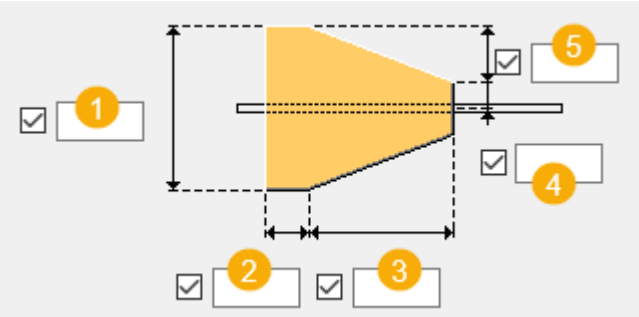
Stiffener length



	Description
1	Distance between the stiffener edge and the gusset plate edge.

	Description
2	Length of stiffener.

Stiffener dimensions

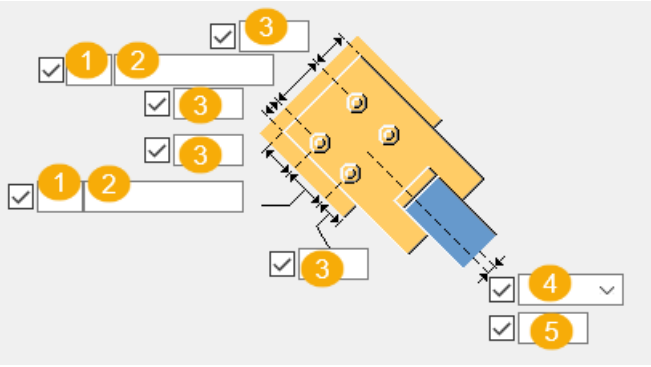


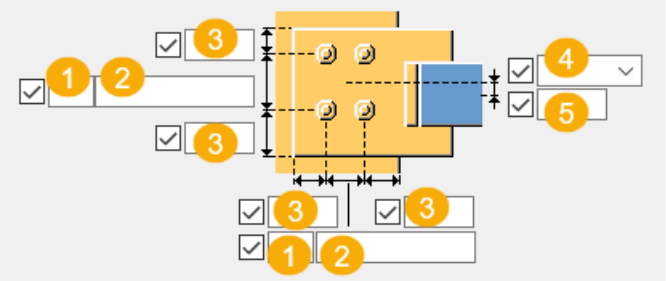
	Description
1	Width of the stiffener.
2	Length of the stiffener base.
3	Length of the skew part of the stiffener.
4	Distance from the stiffener center line.
5	Vertical distance between the stiffener base and the skew part.

Brace bolts 1 / Brace bolts 2 tab

Use the **Brace bolts 1** and **Brace bolts 2** tabs to control the bolts that connect the first and second brace to the gusset plate.

Bolt group dimensions on connection plates

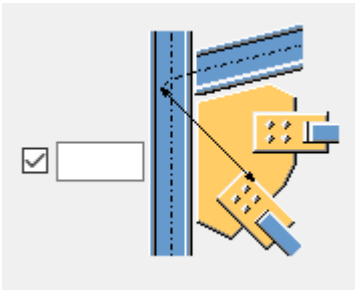
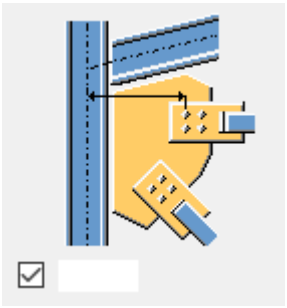




	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Select how to measure the dimensions for vertical bolt group position.
5	Dimension for vertical bolt group position.

Bolt distance

Define the minimum distance from the connection plate bolts to the intersection point of the main part and brace center lines. If a brace is perpendicular to the main part, the distance is measured from the main part center line to the nearest bolts.

Option	Option
	

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt basic properties

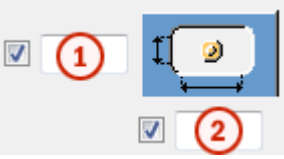
Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Gusset side and brace side bolt type

Select the bolt type to define the location where the bolts should be attached.

Slotted holes

You can define slotted, oversized, or tapped holes.

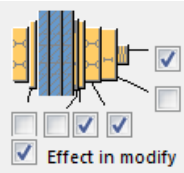


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

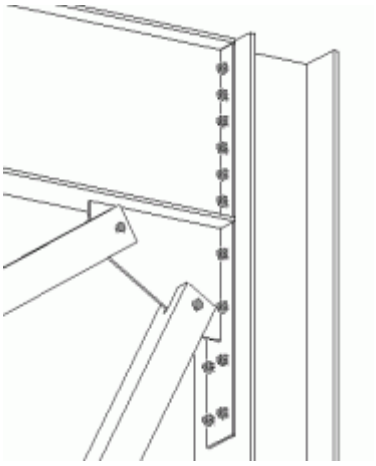
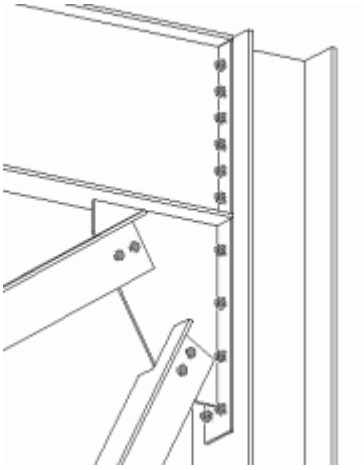
Corner bolted gusset (57)

Corner bolted gusset (57) connects 1 to 10 braces to the corner where two parts meet, using a gusset plate. Welds the gusset plate to the first selected part that forms the corner. Creates optional clip angles, either at the ends of the braces, or on each side. Seals RHS or tube braces.

Objects created

- Gusset plate
- Clip angles (optional)
- Stiffeners
- Brace bolts
- Angle bolts
- Welds

Use for

Situation	Description
	Brace profile: RHS Framing type: Column and extended end plate Gusset plate is welded to an extended end plate. Braces are pinned to the gusset plate.
	Brace profile: T, L Framing type: Column and extended end plate Gusset plate is welded to an extended end plate. Braces are bolted to the gusset plate.

Before you start

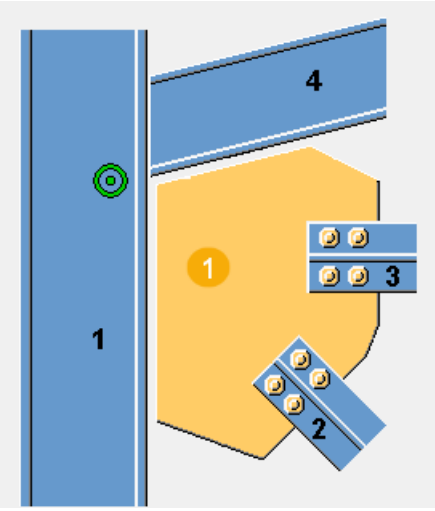
Create two parts to form a corner, and 1 to 10 braces.

Selection order

1. Select the main part (the first part that forms the corner).
   The gusset plate is connected to this part.
2. Select the secondary part (first brace).
3. Select the second secondary part (second brace).
4. Select the subsequent secondary parts (subsequent braces).
5. Select the secondary part that forms the corner.
6. Click the middle mouse button to create the connection.

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

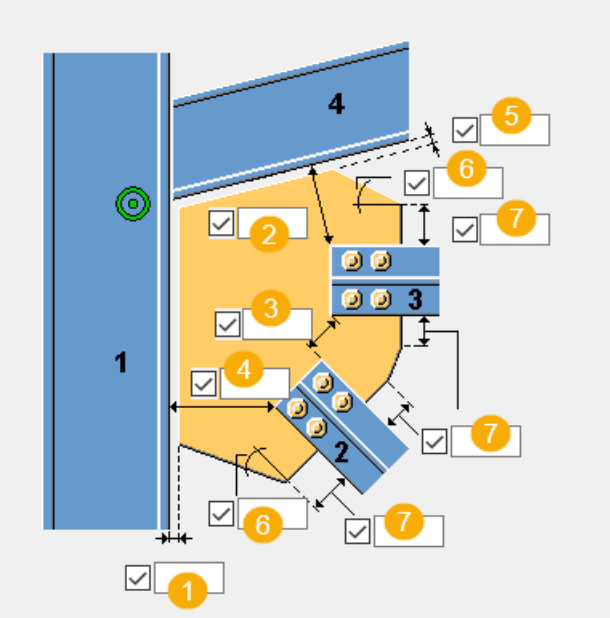
Part identification key



	Description
1	Gusset plate

Picture tab
Use the **Picture** tab to define the gusset plate position and shape.

Dimensions



	Description
1	Define the gap distance between the gusset plate edge and the main part (first part that forms the corner).
2	Define the edge distance between the last secondary part and the last brace.
3	Define the distance between the braces.
4	Define the edge distance between the first brace and the main part.
5	Define the gap distance between the gusset plate edge and the secondary part (second part that forms the corner).
6	Define the gusset plate corner angle (in degrees). This value affects the gusset plate shape.
7	Define the length of the edge of the gusset plate. This value affects the gusset plate shape.

Gusset tab

Use the **Gusset** tab to control the gusset plate properties.

Gusset plate

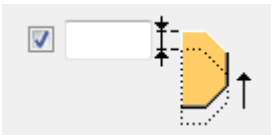
Option	Description
Gusset	Thickness, width and height of the gusset plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

NOTE The following examples show only some of the available options. You will find more options on the **Gusset** tab.

Gusset plate position on the brace

Define where to place the gusset plate on the brace. If needed, you can fine-tune the gusset plate position by moving the gusset plate in the z or in the y direction.

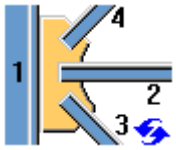
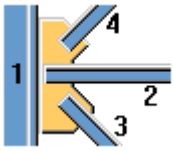
Option	Description
	Default Gusset plate is positioned in the middle of the brace. AutoDefaults can change this option.
	Gusset plate is positioned on the top flange of the brace.
	Define how much the gusset plate is moved in the z direction.
	Define how much the gusset plate is moved in the y direction.

Gusset plate shape

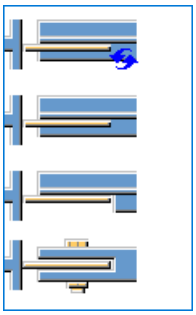
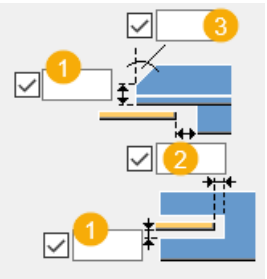
When you select the option to optimize the gusset plate weight, you can define whether the selection order of the braces affects the position of the braces.

Option	Description
	Default AutoDefaults can change this option.
	This option optimizes the gusset plate weight.

Brace position

Option	Description
	Default The brace position is not affected. AutoDefaults can change this option.
	The first selected brace is placed closest to the main part.

Brace notch

Option	Description
	Select whether the brace is notched.
	1. Vertical notch dimension 2. Horizontal notch dimension. 3. Notch angle.

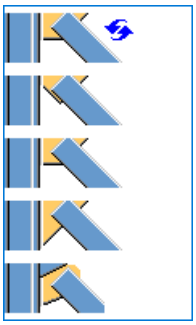
Round cut in bracing

Option	Description
	Default Square cut AutoDefaults can change this option.
	Square cut

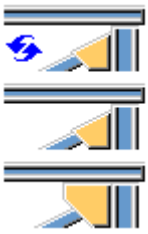
Option	Description
	Round cut Enter the radius value.

Gusset plate shape

The gusset plate edge can be perpendicular either to the main part or the secondary part.

Option	Description
	Select the gusset plate edge shape between the last and second last secondary part.
	Select the gusset plate egde shape between the main part and the first secondary part.

Gusset plate fitting

Option	Description
	Select whether the gusset is fitted to the last selected secondary part.

Gusset welding

Option	Description
	Default Gusset plate is welded to the secondary part. AutoDefaults can change this option.
	Gusset plate is welded to the secondary part.
	Gusset plate is welded to the main part.
	Gusset plate is welded to the main part and the secondary part.

Stiffeners tab

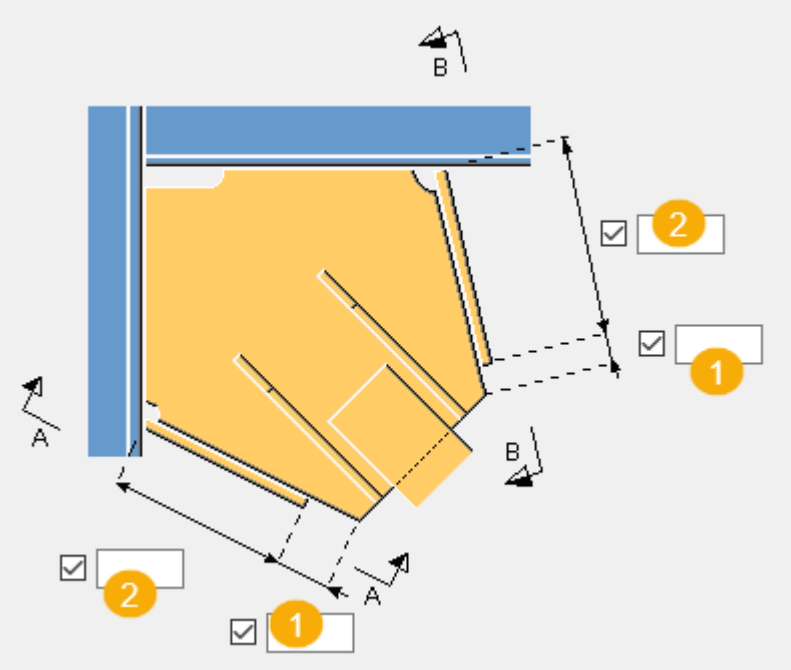
Use the **Stiffeners** tab to control the stiffener properties and dimensions.

Stiffeners

Option	Description
Stiffener 1 **Stiffener 2**	Thickness of the stiffener.

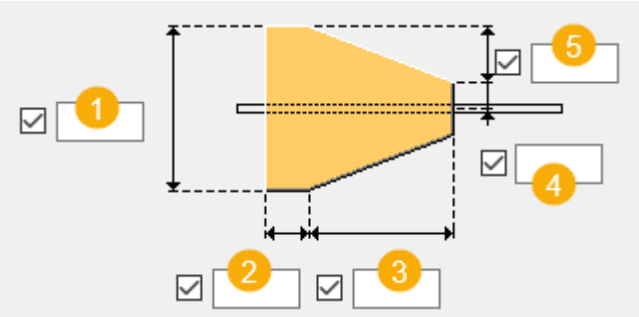
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Stiffener length



	Description
1	Distance between the stiffener edge and the gusset plate edge.
2	Length of stiffener.

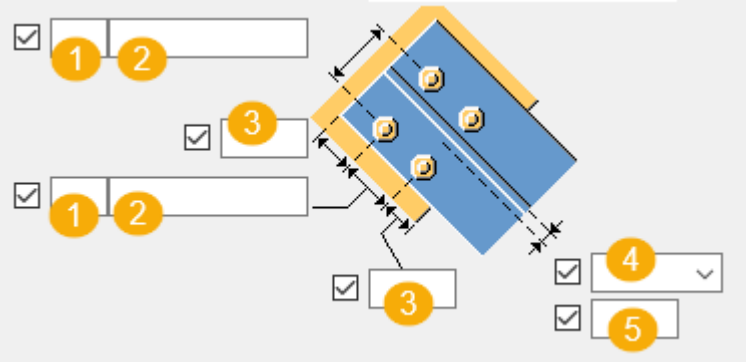
Stiffener dimensions

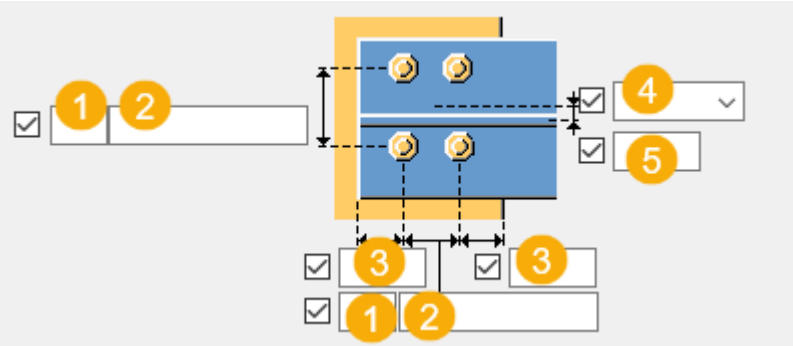


	Description
1	Width of the stiffener.
2	Length of the stiffener base.
3	Length of the skew part of the stiffener.
4	Distance from the stiffener center line.
5	Vertical distance between the stiffener base and the skew part.

Brace bolts 1 / Brace bolts 2 tabs
Use the **Brace bolts 1** and **Brace bolts 2** tabs to control the bolts that connect the first and subsequent braces to the gusset plate.

Bolt group dimensions



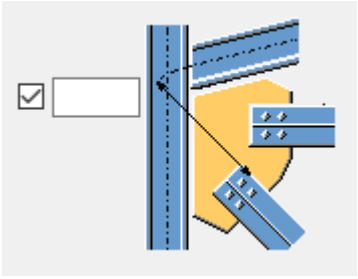
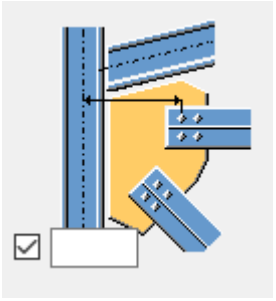


	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Select how to measure the dimensions for vertical bolt group position.
5	Dimension for vertical bolt group position.

Bolt distance

Define the minimum distance from the connection plate bolts to the intersection point of the main part and brace center lines. If a brace is

perpendicular to the main part, the distance is measured from the main part center line to the nearest bolts.

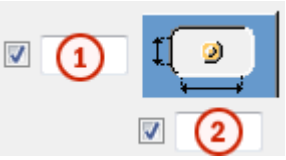
Option	Option
	

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.



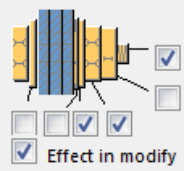
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.

Option	Description	Default
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.

Option	Description
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Angle bolts tab

Use the **Angle bolts** tab to define the clip angles and to control the bolts that connect the clip angle to the gusset plate.

Part

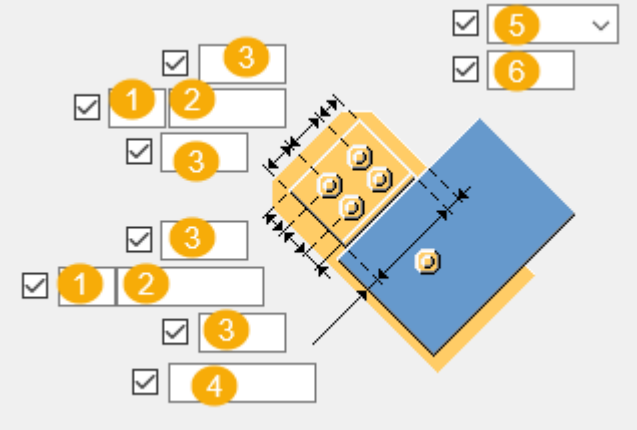
Part	Description
L profile	Select the clip angle profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

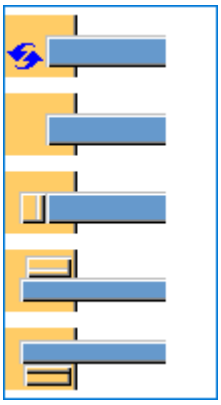
Bolt group dimensions



	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

	Description
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Clip angle edge distance to the brace edge.
5	Select how to measure the dimensions for horizontal bolt group position.
6	Dimension for horizontal bolt group position.

Clip angle position

Option	Description
	Select the clip angle position.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

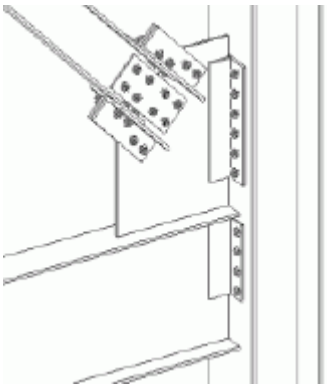
Wraparound gusset (58)

Wraparound gusset (58) connects 1 to 10 braces to the corner where two or three parts meet, usually two beams and a column, using a gusset plate. The gusset plate can be wrapped around the third part, usually a column. Bolts or welds the brace web to the gusset plate using connection plates, and bolts or welds the brace flange to the gusset plate using clip angles. Either connects the gusset plate directly to the two beams, or uses clip angles or shear tabs, or connection plates. The brace profile can be C or W.

Objects created

- Gusset plate
- Clip angles
- Shear tabs
- Connection plates
- Shim plates
- Bolts
- Cuts
- Welds

Use for

Situation	Description
	Brace profile: W Framing type: Beam and column Gusset plate is bolted to the column flange using a clip angle. Braces are bolted to the gusset plate using a connection plate and clip angles.

Before you start

Create 2 or 3 parts that form a corner, and 1 to 10 braces.

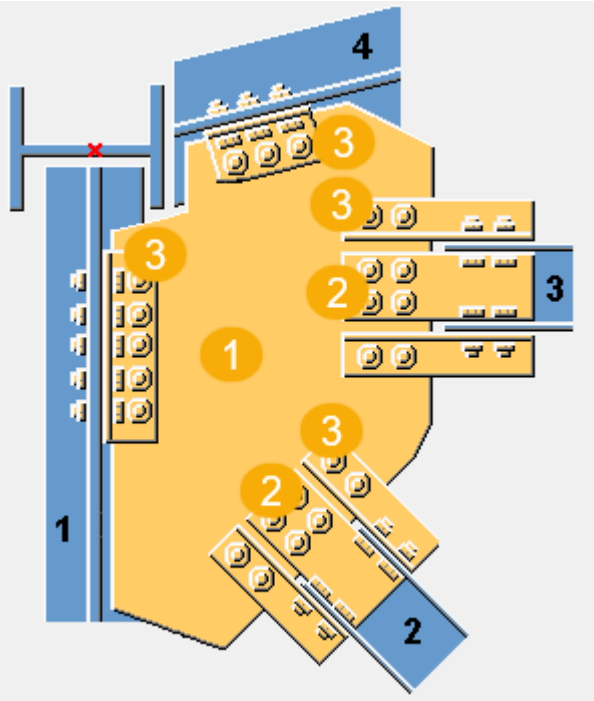
NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Selection order

1. Select the main part (the first part that forms the corner).

2. Select the secondary part (first brace).
3. Select the second secondary part (second brace).
4. Select the subsequent secondary parts (subsequent braces).
5. Select the secondary part that forms the corner.
6. If needed, select the column to wrap the gusset plate around the column where two beams and the column meet.
7. Click the middle mouse button to create the component.

Part identification key

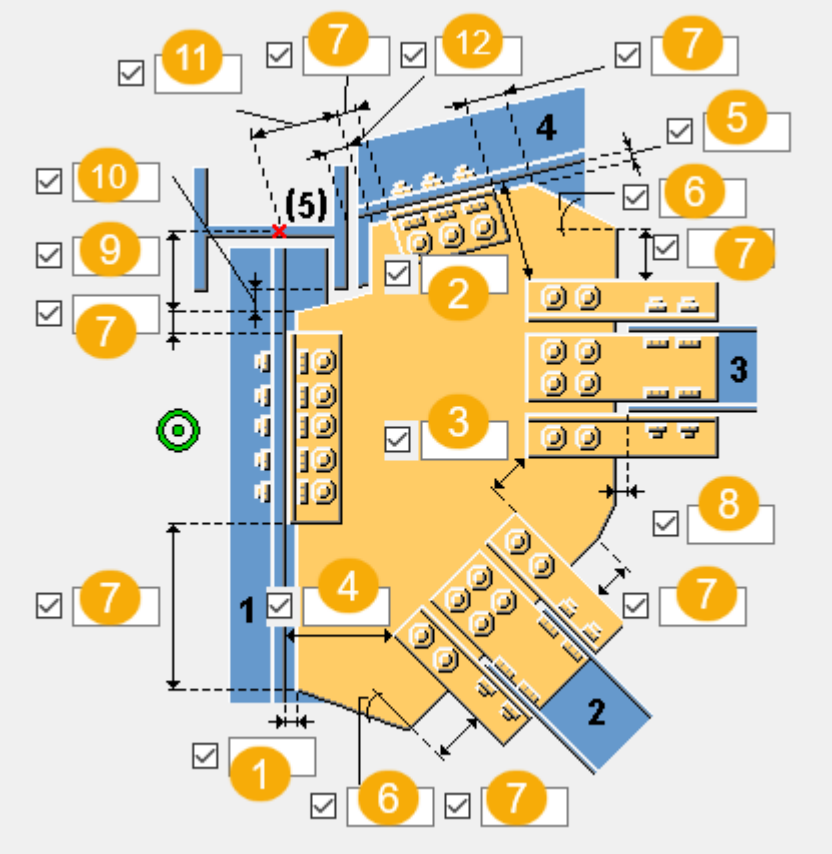


	Description
1	Gusset plate
2	Connection plate
3	Clip angle

Picture tab

Use the **Picture** tab to define the shape of the gusset plate, location of the braces and clip angles, and the work point location.

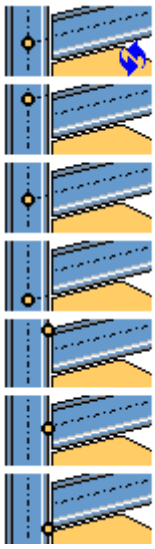
Dimensions



	Description
1	Define the gap distance between the gusset plate edge and the main part.
2	Define the edge distance between the last secondary part and the last brace.
3	Define the distance between the braces.
4	Define the edge distance between the first brace and the main part.
5	Define the gap distance between the gusset plate edge and the secondary part (second part that forms the corner).
6	Define the gusset plate corner angle (in degrees). This value affects the gusset plate shape.
7	Define the length of the edge of the gusset plate. This value affects the gusset plate shape.
8	Define the gap distance between the gusset plate edge and the brace.
9	Define the gusset plate edge distance in relation to the work point.

	Description
10	Define the gusset plate edge distance to the flange of the third part.
11	Define the clip angle edge distance in relation to the work point.
12	Define the clip angle edge distance to the flange of the third part.

Work point position

Option	Description
	Select the work point position. The default position is the point where the two main parts intersect. Tekla Structures uses the work point of a component to calculate check dimensions and part position dimensions in drawings.

Gusset tab

Use the **Gusset** tab to define gusset plate, connection plate and clip angle properties.

Parts

Option	Description
Gusset	Thickness, width and height of the gusset plate.
Connection plates	Thickness and height of the connection plates.
Upper clip angle **Lower clip angle**	Select the clip angle profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Gusset plate connection

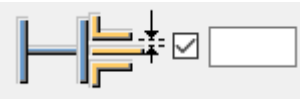
Define how the gusset plate is connected to the main part.

Option	Description
	Default Gusset plate is connected to the main part with a clip angle. AutoDefaults can change this option.
	Gusset plate is connected to the main part with clip angles. Select to which side of the gusset plate the clip angles are created.
	Gusset plate is connected to the main part with a connection plate. Select to which side of the gusset plate the connection plate is created.

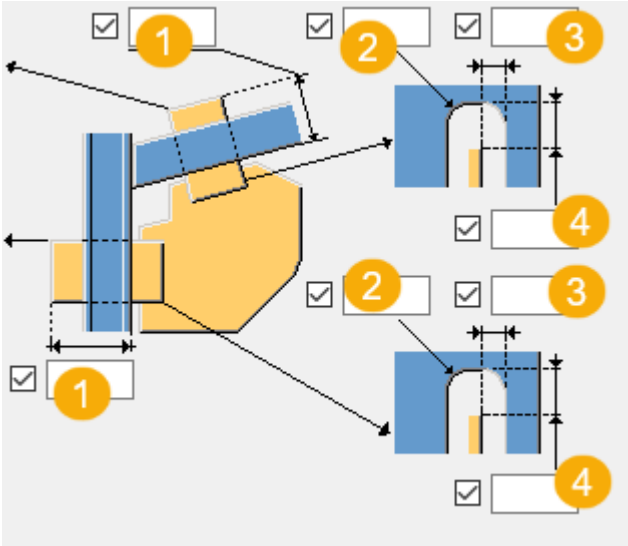
Erection clearance

You can define the clearance for both the main part and the last secondary part connection plates or clip angles.

Gap dimensions

Option	Description
	Define the gap dimension between the gusset plate and the connection plates or the clip angles.

Connection plate cut through dimensions



	Description
1	Connection plate cut through length
2	Chamfer radius
3	Horizontal chamfer dimension
4	Vertical chamfer dimension

Clip angle orientation

Define how the clip angle is placed on the connection.

Option	Description
	Default Clip angle is placed on the connection so that the longer leg is connected to the gusset plate. AutoDefaults can change this option.
	Clip angle is placed on the connection so that the longer leg is connected to the main part.

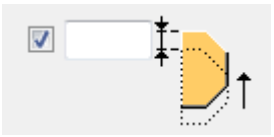
Gusset plate shape

When you select the option to optimize the gusset plate weight, you can define whether the selection order of the braces affects the position of the braces.

Option	Description
	Default AutoDefaults can change this option.
	This option optimizes the gusset plate weight.

Gusset plate position on the brace

Define where to place the gusset plate on the brace. If needed, you can fine-tune the gusset plate position by moving the gusset plate in the z or in the y direction.

Option	Description
	Default Gusset plate is positioned in the middle of the brace. AutoDefaults can change this option.
	Gusset plate is positioned on the top flange of the brace.
	Define how much the gusset plate is moved in the z direction.
	Define how much the gusset plate is moved in the y direction.

Notch angle

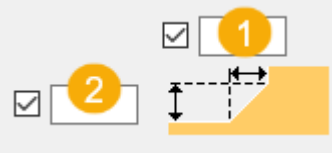
Option	Description
	Default Square notch

Option	Description
	If the gusset plate clashes with the column, the gusset plate is notched to wrap around the column. AutoDefaults can change this option.
	Square notch
	Bevel notch

Notch chamfer type

Option	Description
	Default Bevel chamfer AutoDefaults can change this option.
	No chamfer Notched edges are parallel to the edges of the gusset plate.
	Bevel chamfer
	Rounded chamfer calculated as a quarter of a circle. Enter the diameter of the circle as the horizontal dimension of the chamfer.
	Rounded chamfer calculated as a three quarters of a circle. Enter the diameter of the circle as the horizontal dimension of the chamfer. The center point of the circle is the corner of the notch.
	Corner of the gusset plate is beveled.
	Corner of the gusset plate is not notched or chamfered.

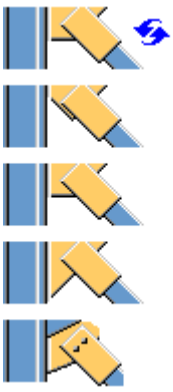
Chamfer dimensions



	Description
1	Horizontal dimension of the chamfer
2	Vertical dimension of the chamfer

Gusset plate shape

The gusset plate edge can be perpendicular either to the main part or the secondary part.

Option	Description
	Select the gusset plate edge shape between the last and second last secondary part.
	Select the gusset plate egde shape between the main part and the first secondary part.

Brace conn tab

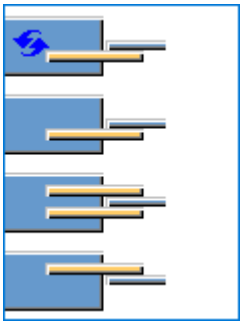
Use the **Brace conn** tab to control the properties of connection plates, clip angles, filler plates, and shear tabs, and the angle connection profile.

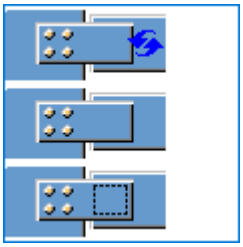
Plates

Option	Description
Connection plate	Thickness and width of the connection plate. Select the connection plate profile.
Upper clip angle **Lower clip angle**	Select the clip angle profile.
Filler plate	Thickness, width and height of the filler plate.
Upper shear tab	Thickness, width and height of the upper shear tab.
Lower shear tab	Height of the lower shear tab.

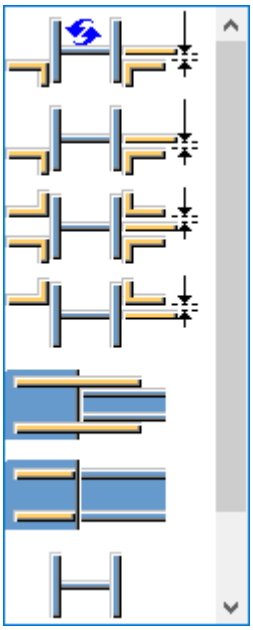
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Plate creation

Option	Description
	Select whether one or two connection plates are created.

Option	Description
	Select whether a filler plate is created between the connection plate and the brace web. The default is that a filler plate is not created.

Clip angle creation

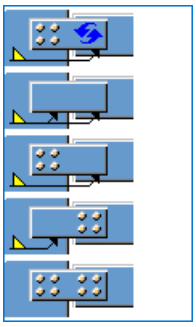
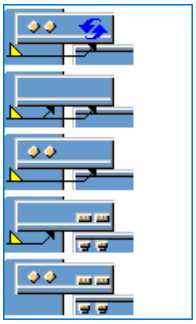
Option	Description
	Define whether the braces are attached to the gusset plate using clip angles or shear tabs, and specify the number of clip angles to create. The default option is to create two clip angles below the brace web.

Clip angle orientation

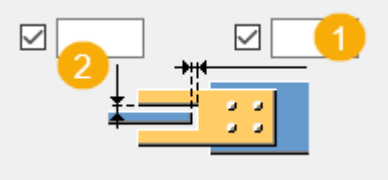
Define how the clip angle is placed on the connection.

Option	Description
	Default Clip angle is placed on the connection so that the longer leg is connected to the gusset plate. AutoDefaults can change this option.
	Clip angle is placed on the connection so that the longer leg is connected to the main part.

Connection type

Option	Description
	Select the connection type (weld or bolts) between the gusset plate and the connection plate.
	Select the connection type (weld or bolt) between the gusset plate and the L profile.

Connection plate gap dimensions



	Description
1	Horizontal gap dimension
2	Vertical gap dimension

Shim plates

Use the **Shim plates** tab tab to define shim plate properties.

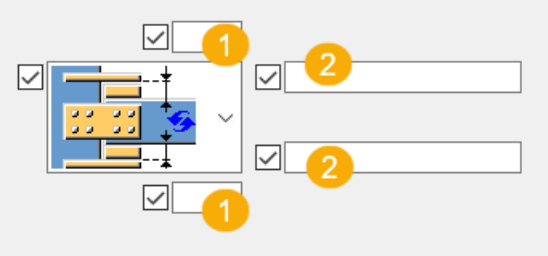
Plates

Option	Description
Shim plate 1 **Shim plate 2** **Shim plate 3**	Thickness, width and height of the shim plates.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Shim plate position

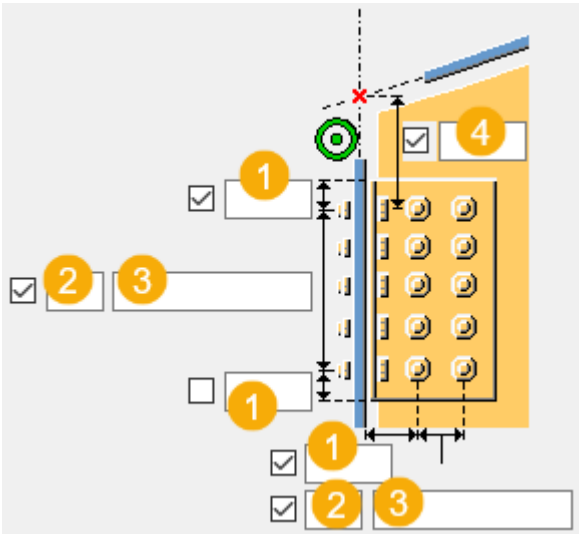
You can create shim plates when connecting braces to the gusset plate using clip angles.

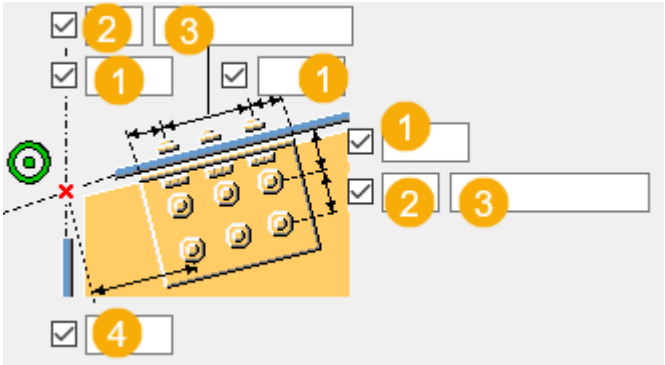


	Description
1	Define the gap between the brace and connection plate.
2	Define how many shim plates are created at the top and bottom flanges. Enter the shim plate profile numbers: 1, 2 or 3. These are the numbers that are on the upper part on the **Shim plates** tab. For example, if you want to create three shim plates at the top flange, and you want to use **Shim plate 1** twice and **Shim plate 1** once, enter 1 1 2. The first number you enter is the shim plate closest to the brace flange.

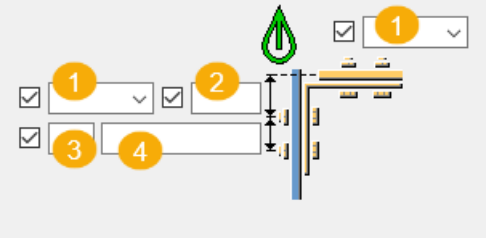
Gusset bolts 1 / Gusset conn 2 tab
Use the **Gusset bolts 1** and **Gusset conn 2** tabs to control the bolt group properties for bolts that connect the gusset plate to the main part and the secondary part, and to control the clip angle attachment.

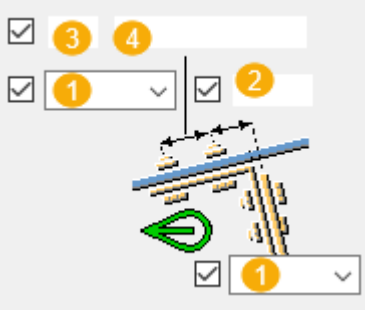
Bolt group dimensions





1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Vertical bolt group dimension in relation to the work point. Work point is the intersection point between the center lines of the main part and the last secondary part.





	Description
1	Location where the bolts should be attached.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

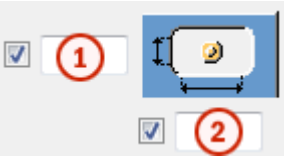
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the	`Yes`

Option	Description	Default
	bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	

Slotted holes

You can define slotted, oversized, or tapped holes.

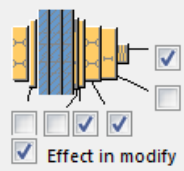


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Clip angle attachment type

Define how the clip angle is attached to the gusset plate and to the main part.

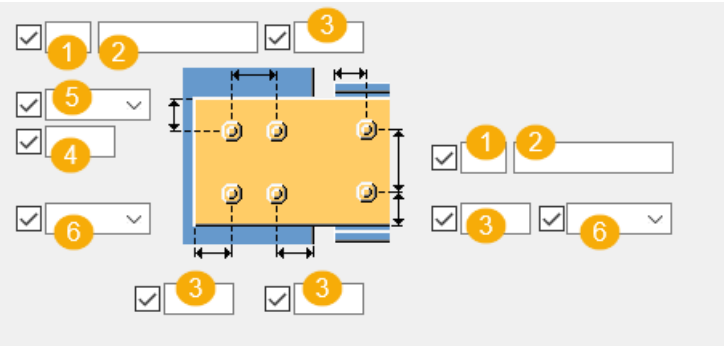
Option	Description
	Default When the main part is a tube profile, the clip angles are welded to the main part and bolted to the secondary part. Otherwise the clip angles are bolted to both parts. AutoDefaults can change this option.
	Automatic When the main part is a tube profile, the clip angles are welded to the main part and bolted to the secondary part. Otherwise the clip angles are bolted to both parts.
	Main part is bolted and secondary part is welded.
	Main part is welded and secondary is part bolted.
	Both parts are bolted.
	Both parts are welded.

Staggering of bolts

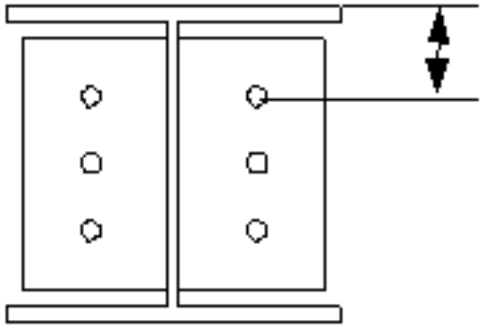
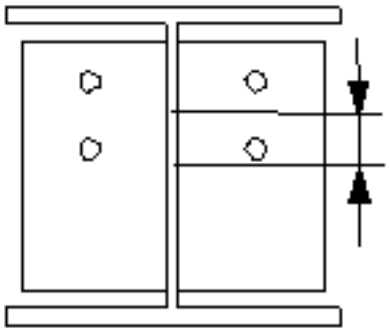
Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

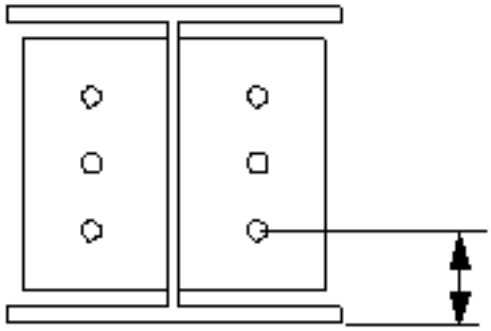
Brace bolts 1 / Brace bolts 2 tab
Use the **Brace bolts 1** and **Brace bolts 2** tabs to control the bolts that connect the braces to the gusset plate.

Bolt group dimensions

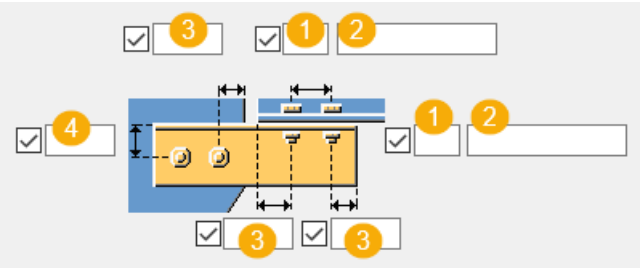


	Option
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

	Option
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Dimension for vertical bolt group position.
5	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt.  • **Middle**: From the center line of the bolts to the center line of the secondary part. 

	Option
	• **Below**: From the lower edge of the secondary part to the lowest bolt. 
6	Select the bolt type.

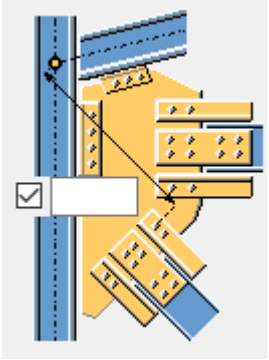
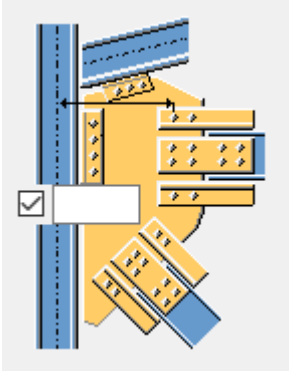
Angle bolt group dimensions



	Option
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Dimension for vertical bolt group position.

Bolt distance

Define the minimum distance from the connection plate bolts to the intersection point of the main part and brace center lines. If a brace is perpendicular to the main part, the distance is measured from the main part center line to the nearest bolts.

Option	Option
	

Vertical bolt position

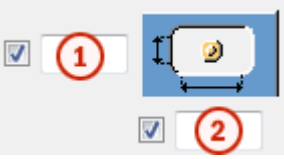
Option	Description
	Bolt position from the L profile edge.
	Bolt position from the secondary part center line.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.

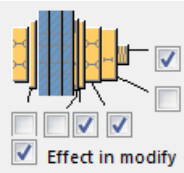


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

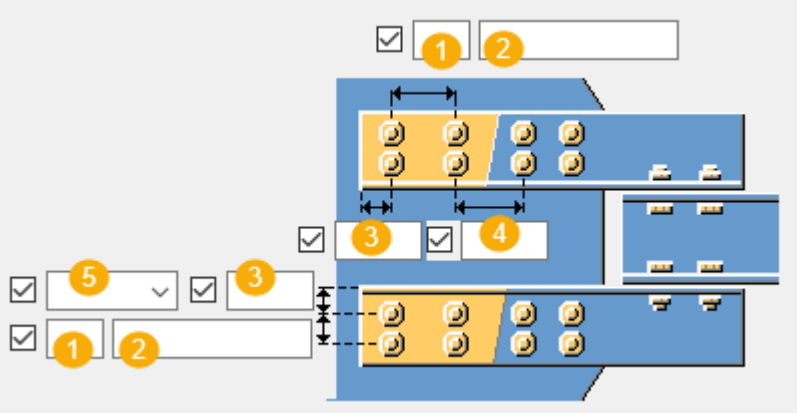
Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Clip angle extra bolts 1 tab
Use the **Clip angle extra bolts 1** tab to define the clip angle bolt group dimensions and bolt properties.

Clip angle extensions

	Select whether clip angles are extended, and the extension sides.

Define the bolt group dimensions of the clip angle extensions.



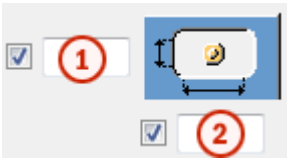
	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Bolt spacing to the bolts in the clip angle extension.
5	Location where the bolts should be attached.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.

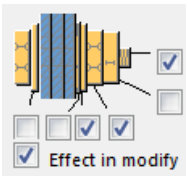


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

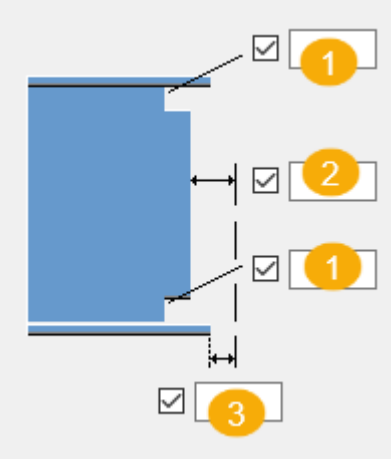
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Beam cut tab

Use the **Beam cut** tab to control weld access holes, beam end preparations, and flange cuts.

Weld access hole dimensions



	Option
1	Dimension for the top and the bottom weld access holes.
2	Gap between the secondary part web and the main part.
3	Gap between the secondary part bottom flange and the main part.

Weld access holes

Option	Option	Description
		Default Round weld access hole AutoDefaults can change this option.
		Round weld access hole
		Square weld access hole

Option	Option	Description
		Diagonal weld access hole

Flange cut

Option for top flange	Option for bottom flange	Description
		Default Flange is not cut. AutoDefaults can change this option.
		Flange is not cut.
		Flange is cut.

Beam end preparation

Option	Description
	Default Top and bottom flange are prepared. AutoDefaults can change this option.
	Automatic Top and bottom flange are prepared.
	Beam end is not prepared.
	Top and bottom flange are prepared.
	Top flange is prepared.
	Bottom flange is prepared.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Create welds

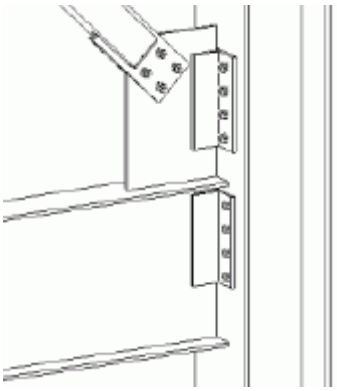
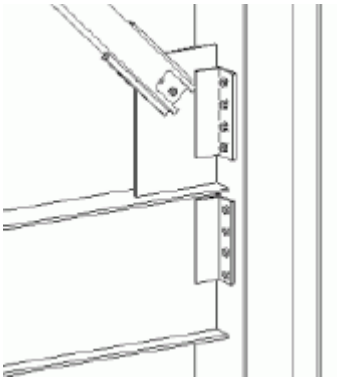
Hollow brace wraparound gusset (59)

Hollow brace wraparound gusset (59) connects 1 to 10 braces to the corner where 2 or 3 parts meet, using a gusset plate. The gusset plate can be wrapped around the third part, usually a column. Bolts or welds the brace web to the gusset plate using connection plates, and bolts or welds the brace flange to the gusset plate using clip angles. Either connects the gusset plate directly to the existing parts, or uses clip angles or connection plates. Seals braces.

Objects created

- Gusset plate
- Connection plates (optional)
- Clip angles (optional)
- Seal plates
- Tongue plates
- Cover plate
- Bolts
- Welds

Use for

Situation	Description
	Brace profile: RHS Framing type: Beam and column Gusset plate is bolted to the column flange using a clip angle. Braces are bolted to the gusset plate using a tongue plate.
	Brace profile: RHS Framing type: Beam and column Gusset plate is bolted to the column flange using a clip angle. Braces are welded to the connection plate. The end of the brace is notched to accommodate the bolts in the connection between the connection plate and the gusset plate.

Before you start

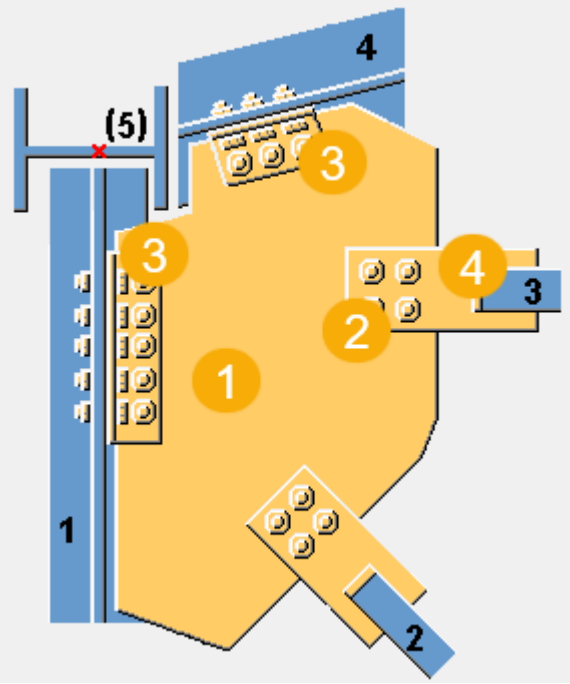
Create 2 or 3 parts that form a corner, and 1 to 10 braces.

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Selection order

1. Select the main part (the first part that forms the corner).
2. Select the secondary part (first brace).
3. Select the second secondary part (second brace).
4. Select the subsequent secondary parts (subsequent braces).
5. Select the secondary part that forms the corner (Tekla Structures connects the gusset plate to this part).
6. If needed, select the column to wrap the gusset plate around the column where two beams and the column meet.
7. Click the middle mouse button to create the connection.

Part identification key

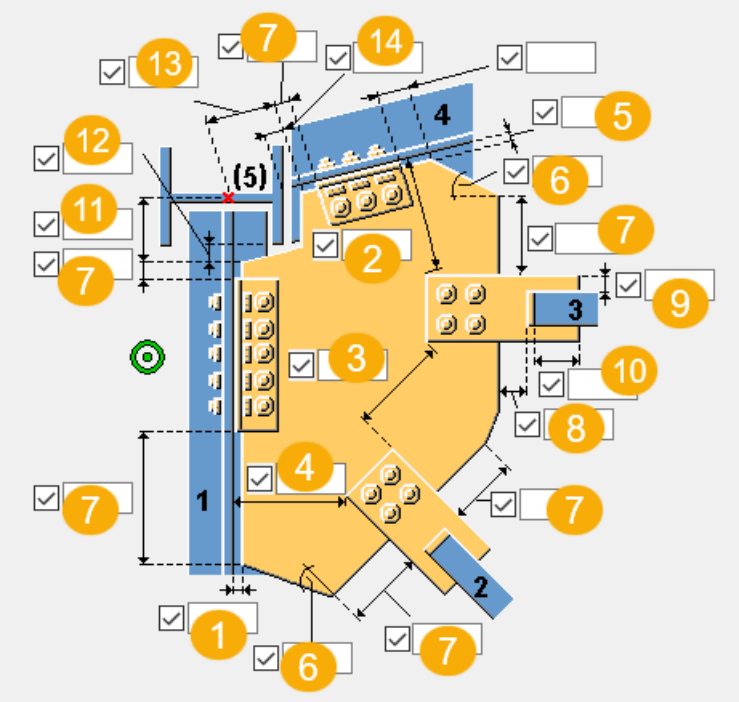


	Description
1	Gusset plate
2	Connection plate
3	Clip angle
4	Seal plate

Picture tab

Use the **Picture** tab to define the shape of the gusset plate, location of the braces and clip angles, and the work point location.

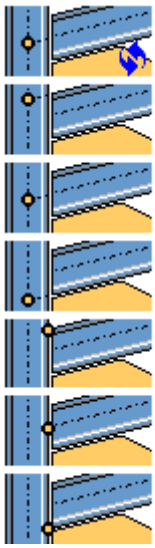
Dimensions



	Description
1	Define the gap distance between the gusset plate edge and the main part.
2	Define the edge distance between the last secondary part and the last brace.
3	Define the distance between the braces.
4	Define the edge distance between the first brace and the main part.
5	Define the gap distance between the gusset plate edge and the secondary part (second part that forms the corner).
6	Define the gusset plate corner angle (in degrees). This value affects the gusset plate shape.
7	Define the length of the edge of the gusset plate. This value affects the gusset plate shape.
8	Define the gap distance between the gusset plate edge and the seal plate.
9	Define the length of the edge of the connection plate.
10	Define the brace dimension on the connection plate.
11	Define the gusset plate edge distance in relation to the work point.

	Description
12	Define the gusset plate edge distance to the flange of the third part.
13	Define the clip angle edge distance in relation to the work point.
14	Define the clip angle edge distance to the flange of the third part.

Work point position

Option	Description
	Select the work point position. The default position is the point where the two main parts intersect. Tekla Structures uses the work point of a component to calculate check dimensions and part position dimensions in drawings.

Gusset tab

Use the **Gusset** tab to define the gusset plate, connection plate and clip angle properties.

Parts

Option	Description
Gusset	Thickness, width and height of the gusset plate.
Connection plates	Thickness and height of the connection plates.
Upper clip angle **Lower clip angle**	Select the clip angle profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

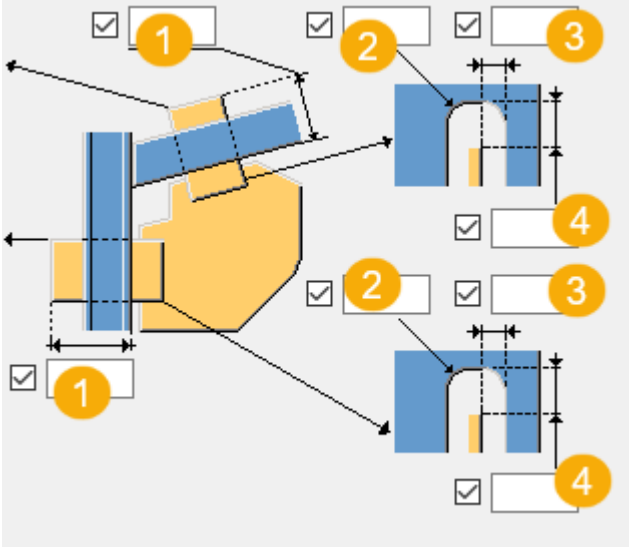
NOTE The following examples show only some of the available options. You will find more options on the **Gusset** tab.

Gusset plate connection

Define how the gusset plate is connected to the main part.

Option	Description
	Default Gusset plate is connected to the main part with a clip angle. AutoDefaults can change this option.
	Gusset plate is connected to the main part with clip angles. Select to which side of the gusset plate the clip angles are created.
	Gusset plate is connected to the main part with a connection plate. Select to which side of the gusset plate the connection plate is created.

Connection plate cut through dimensions



	Description
1	Connection plate cut through length
2	Chamfer radius
3	Horizontal chamfer dimension
4	Vertical chamfer dimension

Clip angle orientation

Define how the clip angle is placed on the connection.

Option	Description
	Default Clip angle is placed on the connection so that the longer leg is connected to the gusset plate. AutoDefaults can change this option.
	Clip angle is placed on the connection so that the longer leg is connected to the main part.

Erection clearance

You can define the clearance for both the main part and the last secondary part connection plates or clip angles.

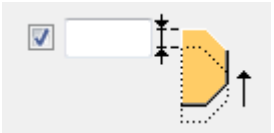
Gusset plate shape

When you select the option to optimize the gusset plate weight, you can define whether the selection order of the braces affects the position of the braces.

Option	Description
	Default AutoDefaults can change this option.
	This option optimizes the gusset plate weight.

Gusset plate position on the brace

Define where to place the gusset plate on the brace. If needed, you can fine-tune the gusset plate position by moving the gusset plate in the z or in the y direction.

Option	Description
	Default Gusset plate is positioned in the middle of the brace. AutoDefaults can change this option.
	Gusset plate is positioned on the top flange of the brace.
	Define how much the gusset plate is moved in the z direction.
	Define how much the gusset plate is moved in the y direction.

Notch angle

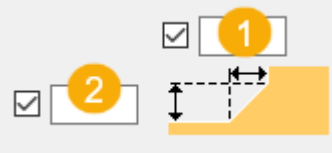
Option	Description
	Default Square notch If the gusset plate clashes with the column, the gusset plate is notched to wrap around the column. AutoDefaults can change this option.
	Square notch

Option	Description
	Bevel notch

Notch chamfer type

Option	Description
	Default Bevel chamfer AutoDefaults can change this option.
	No chamfer Notched edges are parallel to the edges of the gusset plate.
	Bevel chamfer
	Rounded chamfer calculated as a quarter of a circle. Enter the diameter of the circle as the horizontal dimension of the chamfer.
	Rounded chamfer calculated as a three quarters of a circle. Enter the diameter of the circle as the horizontal dimension of the chamfer. The center point of the circle is the corner of the notch.
	Corner of the gusset plate is beveled.
	Corner of the gusset plate is not notched or chamfered.

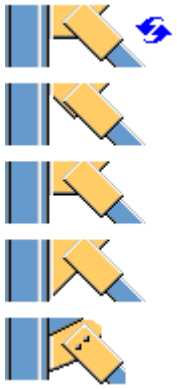
Chamfer dimensions



	Description
1	Horizontal dimension of the chamfer
2	Vertical dimension of the chamfer

Gusset plate shape

The gusset plate edge can be perpendicular either to the main part or the secondary part.

Option	Description
	Select the gusset plate edge shape between the last and second last secondary part.
	Select the gusset plate egde shape between the main part and the first secondary part.

Brace conn tab

Use the **Brace conn** tab to control the connection plate, seal plate, tongue plate, and cover plate properties.

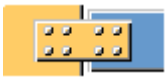
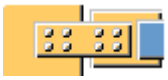
Parts

Option	Description
Connection plate	Thickness, width and height of the connection plate.
Seal plates	Thickness and height of the seal plates.
Tongue plate	Thickness and height of the tongue plate.
Cover plate	Thickness, width and height of the cover plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Brace connection types

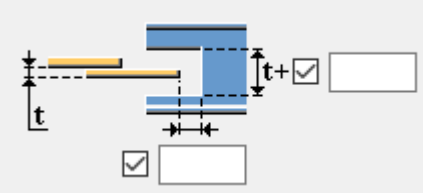
Define how the brace is connected to the connection plate.

Option	Description
	Default Brace is welded. AutoDefaults can change this option.
	Brace is welded.
	Brace is bolted.
	Brace is welded and notched around the nuts.
	Tongue plate and cover plate are created.

Cut in bracing

Define the width of the cut in the bracing, where **t** is the thickness of the connection plate.

Define the length of the cut in the bracing from the edge of the connection plate.



Round cut in bracing

Option	Description
	Default Square cut AutoDefaults can change this option.
	Square cut
	Round cut Enter the radius value.

Connection plate

Define whether the brace is notched or the connection plate cut when the connection plate is connected to the brace.

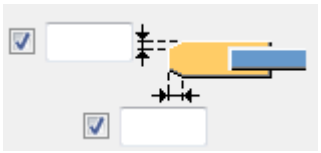
Option	Description
	Default Brace is notched. AutoDefaults can change this option.
	Brace is notched.
	Connection plate is cut.
	If you cut the connection plate, you can define the size of the gap between the brace and the connection plate.

Number of connection plates

Define whether one or two connection plates are used for connecting the brace to the gusset plate.

Option	Description
	Default One connection plate. AutoDefaults can change this option.
	One connection plate.
	Two connection plates

Connection plate chamfer

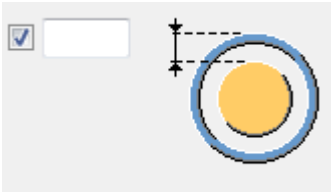
Option	Description
	Default No chamfers are created. AutoDefaults can change this option.
	No chamfers are created.
	Chamfers are created.
	If you create chamfers, define the vertical and horizontal chamfer dimensions.

End plates

If you use the end plates to seal the braces, define the end plate shape and dimensions.

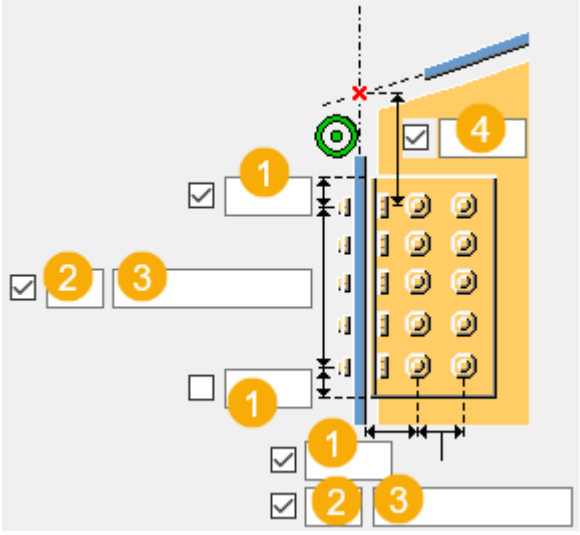
Option	Description
	Default Square end plate. AutoDefaults can change this option.
	Square end plate.
	Round end plate.

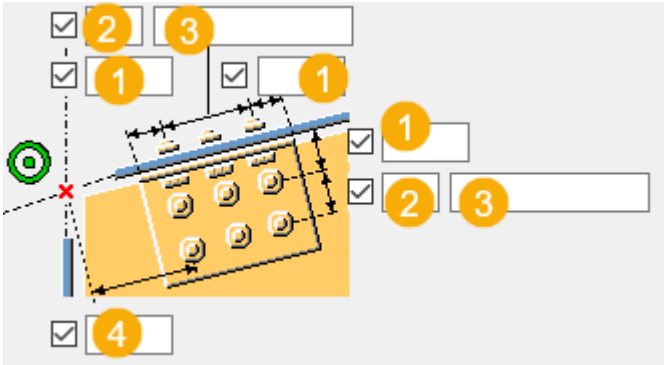
End plate edge distance

Option	Description
	End plate edge distance from the brace outer edge.

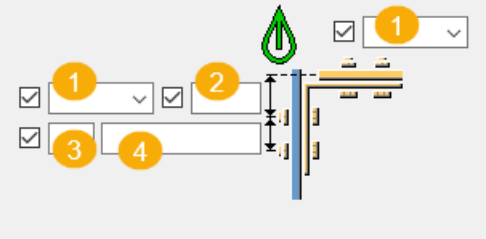
Gusset conn 1 / Gusset conn 2 tab
Use the **Gusset conn 1** and **Gusset conn 2** tabs to control the bolt group properties for bolts that connect the gusset plate to the main part and secondary parts, and to control the clip angle attachment.

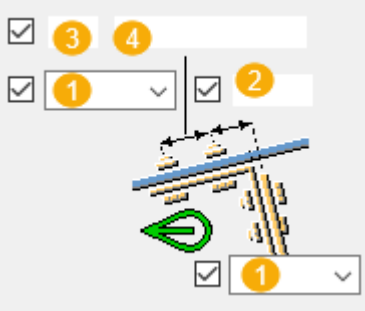
Bolt group dimensions





	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Vertical bolt group dimension in relation to the work point.





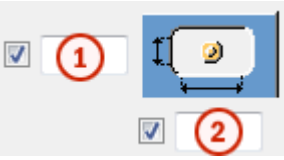
1	Location where the bolts should be attached.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.



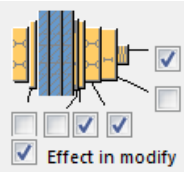
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options	

Option	Description	Default
	depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

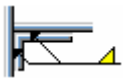
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Clip angle attachment type

Define how the clip angle is attached to the gusset plate and to the main part.

Option	Description
	Default When the main part is a tube profile, the clip angles are welded to the main part and bolted to the secondary part. Otherwise the clip angles are bolted to both parts. AutoDefaults can change this option.
	Automatic When the main part is a tube profile, the clip angles are welded to the main part and bolted to the secondary part. Otherwise the clip angles are bolted to both parts.
	Main part is bolted and secondary part is welded.

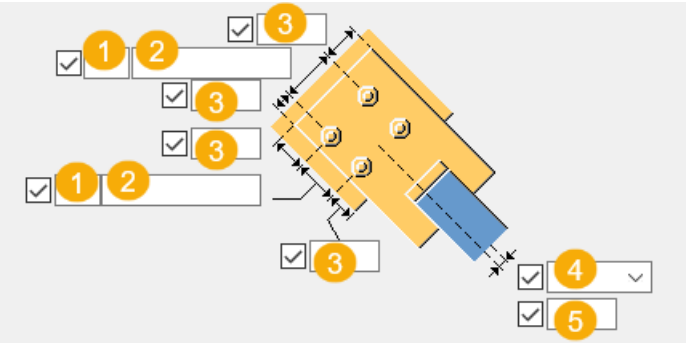
Option	Description
	Main part is welded and secondary is part bolted.
	Both parts are bolted.
	Both parts are welded.

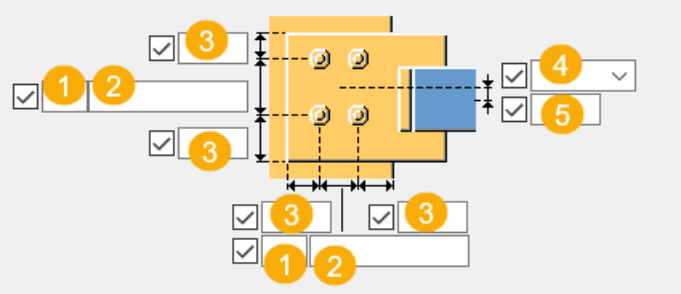
Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Brace bolts 1 / Brace bolts 2 tab
Use the **Brace bolts 1** and **Brace bolts 2** tabs to control the bolts that connect the first and second brace to the gusset plate.

Bolt group dimensions on connection plates

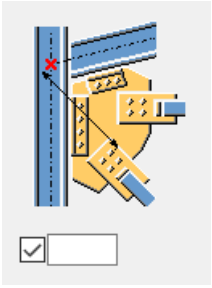
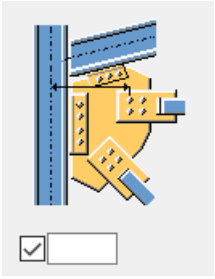




	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Select how to measure the dimensions for vertical bolt group position.
5	Dimension for vertical bolt group position.

Bolt distance

Define the minimum distance from the connection plate bolts to the intersection point of the main part and brace center lines. If a brace is perpendicular to the main part, the distance is measured from the main part center line to the nearest bolts.

Option	Option
	

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Bolt basic properties

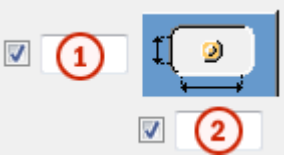
Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Gusset side and brace side bolt type

Select the bolt type to define the location where the bolts should be attached.

Slotted holes

You can define slotted, oversized, or tapped holes.

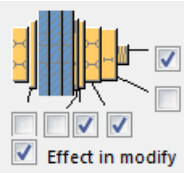


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

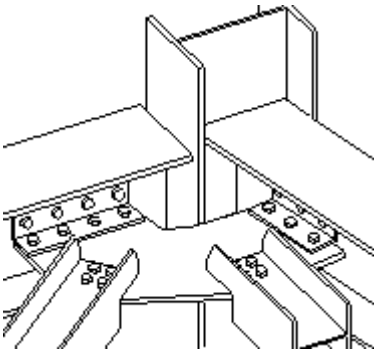
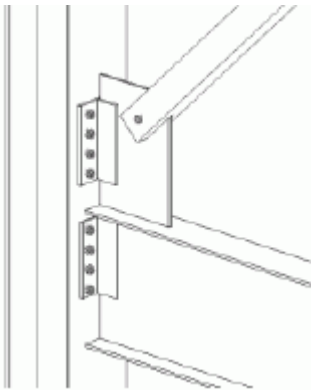
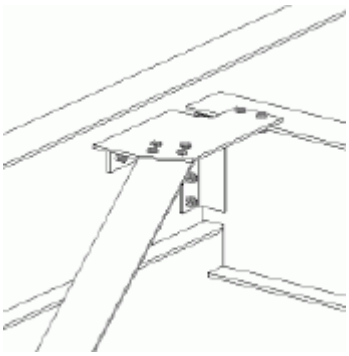
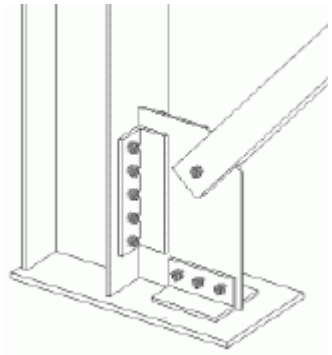
Wraparound gusset cross (60)

Wraparound gusset cross (60) connects 1 to 10 braces to a gusset plate where 2 or 3 parts meet to form a corner, using clip angles and connection plates. The gusset plate can be wrapped around the third part, usually a column. Connects the gusset plate to the parts using clip angles or connection plates, or welds it directly to the first selected part. Clip angles can be created, either at the ends of the braces, or on each side. Seals hollow braces. The braces can have L, W, WT, RHS, and tube profiles.

Objects created

- Gusset plate
- Connection plates
- Clip angles (optional)
- Seal plates (hollow braces)
- Welds

Use for

Situation	Description
	Brace profile: W Framing profile: Column and 2 beams Gusset plate is bolted to the beams using clip angles. Braces are notched and bolted to the gusset plate.
	Brace profile: RHS Framing profile: Column and beam Gusset plate is bolted to the column using a clip angle. Brace is pinned to the gusset plate.
	Brace profile: L Framing profile: Two beams with different elevations Gusset plate is notched and bolted to both beams using clip angles. Brace is bolted to the gusset plate.
	Brace profile: RHS Framing profile: Column and column base plate Gusset plate is bolted to the column and the base plate using clip angles. Brace is pinned to the gusset plate.

Before you start

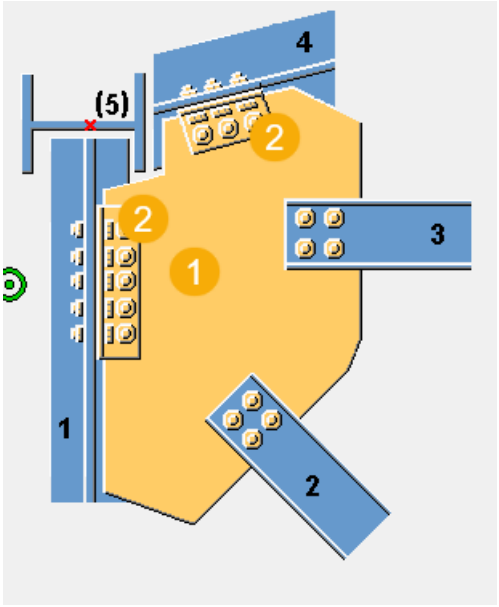
Create 2 or 3 parts that meet to form a corner, and 1 to 10 braces.

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Selection order

1. Select the main part (the first part that forms the corner).
2. Select the secondary part (first brace).
3. Select the second secondary part (second brace).
4. Select the subsequent secondary parts (subsequent braces).
5. Select the secondary part that forms the corner.
6. f needed, select the column to wrap the gusset plate around the column where two beams and the column meet.
7. Click the middle mouse button to create the component.

Part identification key

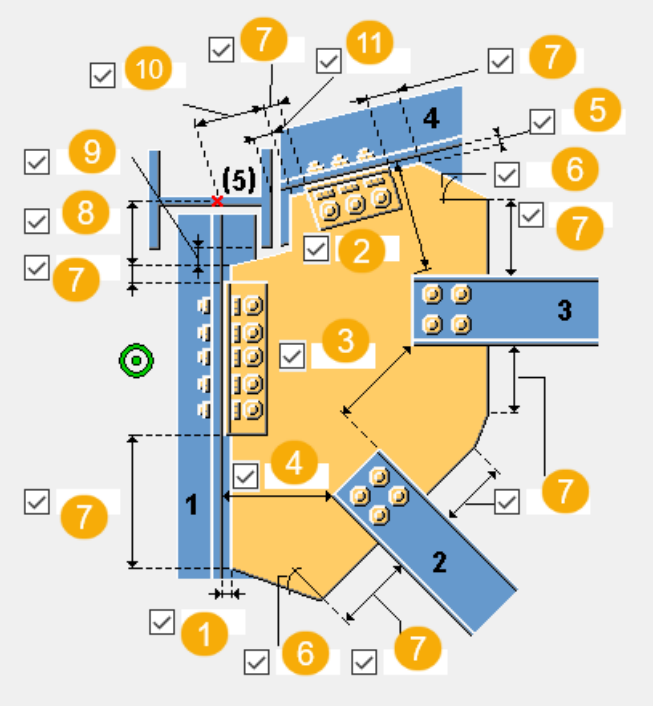


	Description
1	Gusset plate
2	Clip angle

Picture tab

Use the **Picture** tab to define the shape of the gusset plate, location of the braces and clip angles, and the work point location.

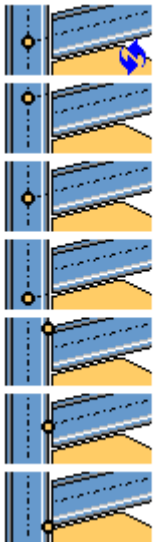
Dimensions



	Description
1	Define the gap distance between the gusset plate edge and the main part.
2	Define the edge distance between the last secondary part and the last brace.
3	Define the distance between the braces.
4	Define the edge distance between the first brace and the main part.
5	Define the gap distance between the gusset plate edge and the secondary part (second part that forms the corner).
6	Define the gusset plate corner angle (in degrees). This value affects the gusset plate shape.
7	Define the length of the edge of the gusset plate. This value affects the gusset plate shape.
8	Define the gusset plate edge distance in relation to the work point.
9	Define the gusset plate edge distance to the flange of the third part.
10	Define the clip angle edge distance in relation to the work point.

	Description
11	Define the clip angle edge distance to the flange of the third part.

Work point position

Option	Description
	Select the work point position. The default position is the point where the two main parts intersect. Tekla Structures uses the work point of a component to calculate check dimensions and part position dimensions in drawings.

Gusset tab

Use the **Gusset** tab to define the gusset plate, connection plate and clip angle properties.

Parts

Option	Description
Gusset	Thickness, width and height of the gusset plate.
Connection plates	Thickness and width of the connection plates.
Upper clip angle **Lower clip angle**	Select the clip angle profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

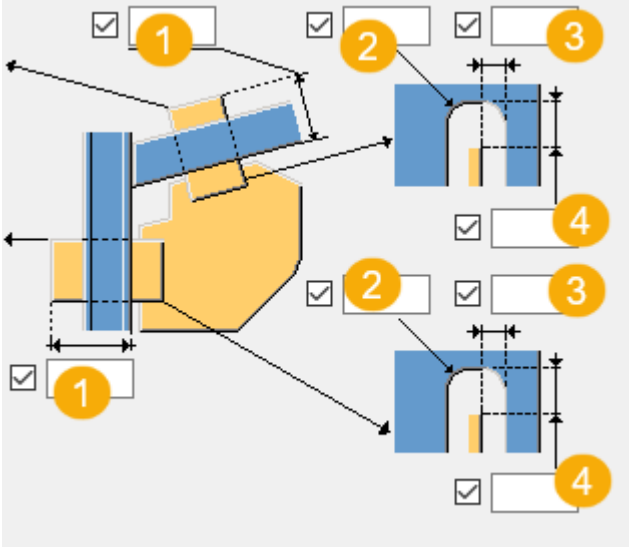
Option	Description	Default
	assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Finish	Describes how the part surface has been treated.	

Gusset plate connection

Define how the gusset plate is connected to the main part.

Option	Description
	Default Gusset plate is connected to the main part with a clip angle. AutoDefaults can change this option.
	Gusset plate is connected to the main part with clip angles. Select to which side of the gusset plate the clip angles are created.
	Gusset plate is connected to the main part with a connection plate. Select to which side of the gusset plate the connection plate is created.

Connection plate cut through dimensions



	Description
1	Connection plate cut through length
2	Chamfer radius
3	Horizontal chamfer dimension
4	Vertical chamfer dimension

Clip angle orientation

Define how the clip angle is placed on the connection.

Option	Description
	Default Clip angle is placed on the connection so that the longer leg is connected to the gusset plate. AutoDefaults can change this option.
	Clip angle is placed on the connection so that the longer leg is connected to the main part.

Erection clearance

You can define the clearance for both the main part and the last secondary part connection plates or clip angles.

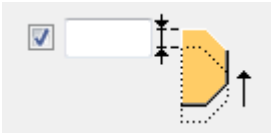
Gusset plate shape

When you select the option to optimize the gusset plate weight, you can define whether the selection order of the braces affects the position of the braces.

Option	Description
	Default AutoDefaults can change this option.
	This option optimizes the gusset plate weight.

Gusset plate position on the brace

Define where to place the gusset plate on the brace. If needed, you can fine-tune the gusset plate position by moving the gusset plate in the z or in the y direction.

Option	Description
	Default Gusset plate is positioned in the middle of the brace. AutoDefaults can change this option.
	Gusset plate is positioned on the top flange of the brace.
	Define how much the gusset plate is moved in the z direction.
	Define how much the gusset plate is moved in the y direction.

Notch angle

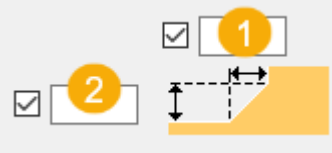
Option	Description
	Default Square notch If the gusset plate clashes with the column, the gusset plate is notched to wrap around the column. AutoDefaults can change this option.
	Square notch

Option	Description
	Bevel notch

Notch chamfer type

Option	Description
	Default Bevel chamfer AutoDefaults can change this option.
	No chamfer Notched edges are parallel to the edges of the gusset plate.
	Bevel chamfer
	Rounded chamfer calculated as a quarter of a circle. Enter the diameter of the circle as the horizontal dimension of the chamfer.
	Rounded chamfer calculated as a three quarters of a circle. Enter the diameter of the circle as the horizontal dimension of the chamfer. The center point of the circle is the corner of the notch.
	Corner of the gusset plate is beveled.
	Corner of the gusset plate is not notched or chamfered.

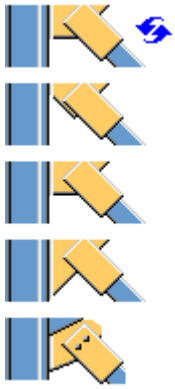
Chamfer dimensions



	Description
1	Horizontal dimension of the chamfer
2	Vertical dimension of the chamfer

Gusset plate shape

The gusset plate edge can be perpendicular either to the main part or the secondary part.

Option	Description
	Select the gusset plate edge shape between the last and second last secondary part.
	Select the gusset plate egde shape between the main part and the first secondary part.

Brace conn tab

Use the **Brace conn** tab to define the seal plate, brace notch and slot properties.

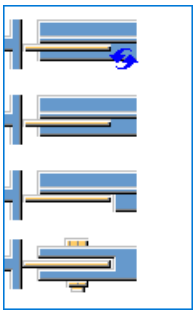
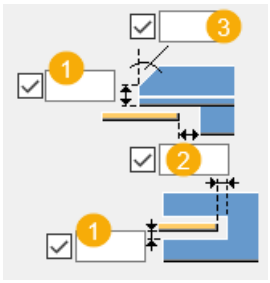
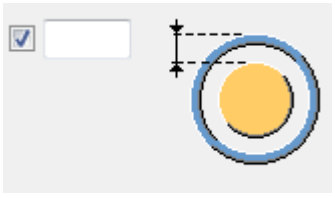
Part

Option	Description
Seal plate	Thickness of the seal plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.

Option	Description	Default
	assembly position number.	
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Brace notch

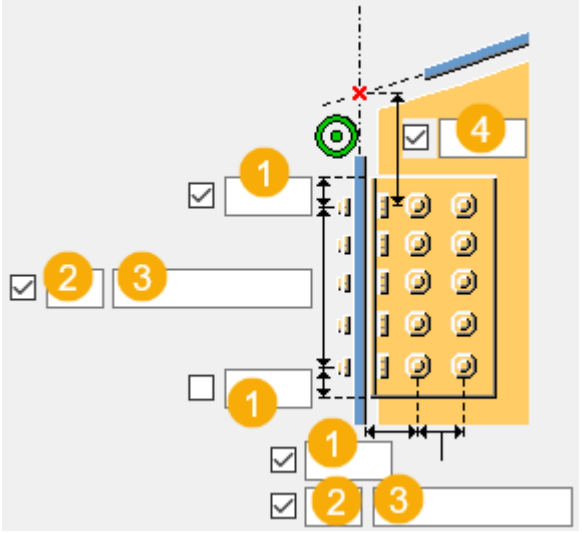
Option	Description
	Select whether the brace is notched.
	1. Vertical notch dimension. 2. Horizontal notch dimension. 3. Notch angle.
	Select whether the secondary parts (except the last secondary part) are fitted.
	Seal plate edge distance from the brace outer edge.

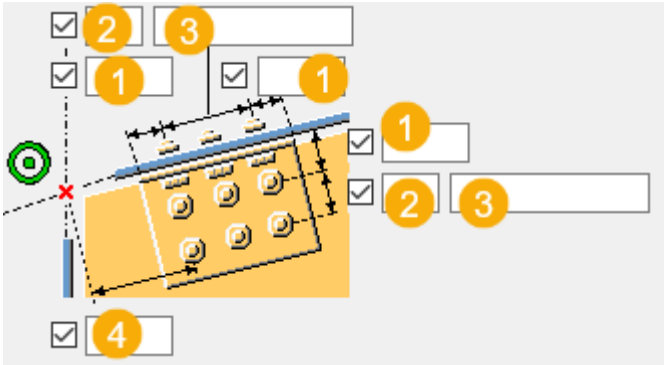
Round cut in bracing

Option	Description
	Default Square cut AutoDefaults can change this option.
	Square cut
	Round cut Enter the radius value.

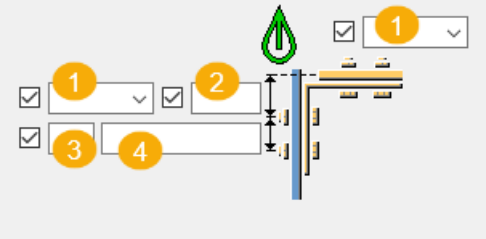
Gusset conn 1 / Gusset conn 2 tabs
Use the **Gusset conn 1** and **Gusset conn 2** tabs to control the bolt group properties for bolts that connect the gusset plate to the main and secondary parts, and to control the clip angle attachment.

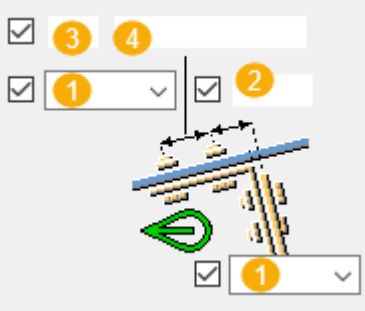
Bolts group dimensions





	Description
1	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Vertical bolt group dimension in relation to the work point.





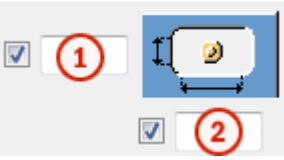
	Description
1	Location where the bolts should be attached.
2	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
3	Number of bolts.
4	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.



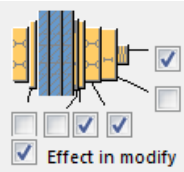
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options	

Option	Description	Default
	depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

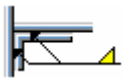
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Clip angle attachment type

Define how the clip angle is attached to the gusset plate and to the main part.

Option	Description
	Default When the main part is a tube profile, the clip angles are welded to the main part and bolted to the secondary part. Otherwise the clip angles are bolted to both parts. AutoDefaults can change this option.
	Automatic When the main part is a tube profile, the clip angles are welded to the main part and bolted to the secondary part. Otherwise the clip angles are bolted to both parts.
	Main part is bolted and secondary part is welded.

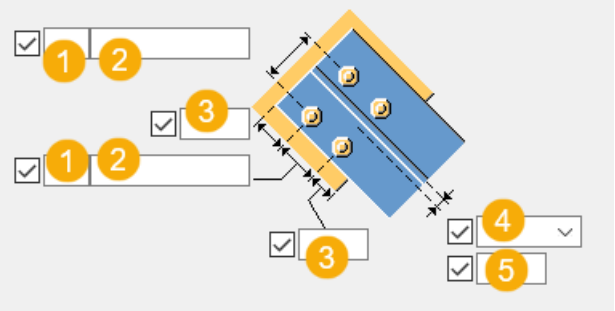
Option	Description
	Main part is welded and secondary is part bolted.
	Both parts are bolted.
	Both parts are welded.

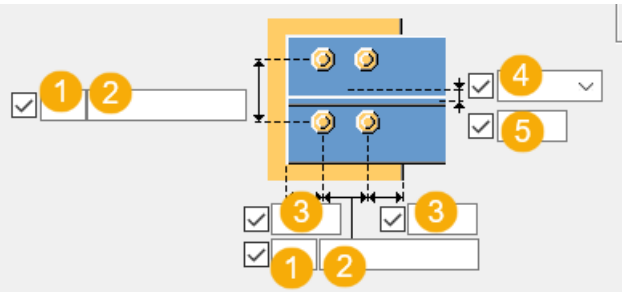
Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Brace bolts 1 / Brace bolts 2 tab
Use the **Brace bolts 1** and **Brace bolts 2** tabs to control the bolts that connect the first and second brace to the gusset plate.

Bolt group dimensions

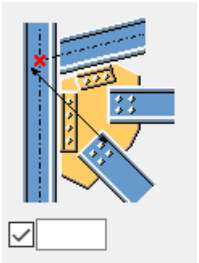
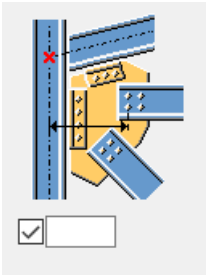




	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Select how to measure the dimensions for vertical bolt group position.
5	Dimension for vertical bolt group position.

Bolt distance

Define the minimum distance from the connection plate bolts to the intersection point of the main part and brace center lines. If a brace is perpendicular to the main part, the distance is measured from the main part center line to the nearest bolts.

Option	Option
	

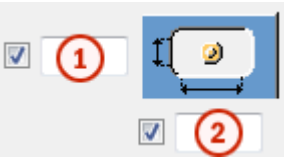
Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.

Option	Description	Default
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

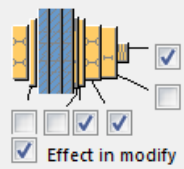


Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2
	Staggered type 3
	Staggered type 4

Angle bolts tab

Use the **Angle bolts** tab to control the bolts that connect the clip angles.

Part

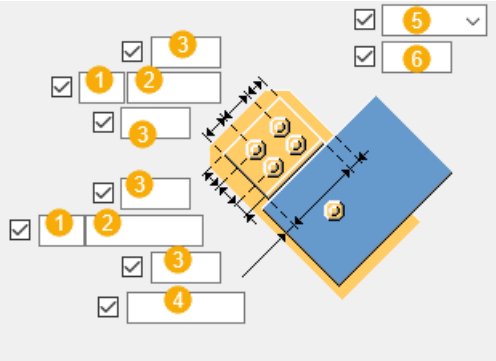
Option	Description
L profile	Select the clip angle profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Bolt basic properties

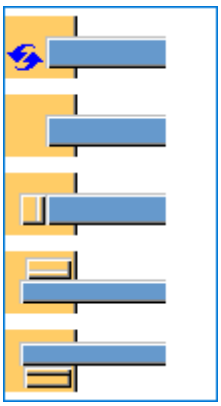
Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Bolt group dimensions



	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Define the edge distance between the clip angle and the brace.
5	Select how to measure the dimensions for vertical bolt group position.
6	Dimension for vertical bolt group position.

Clip angle position

Option	Description
	Select the clip angle position.

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

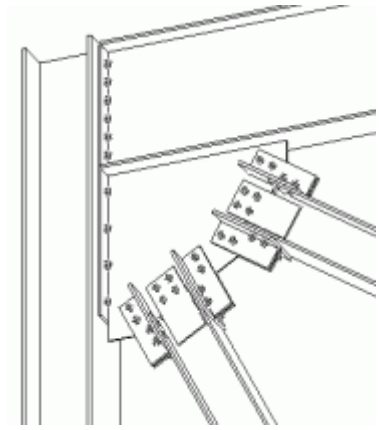
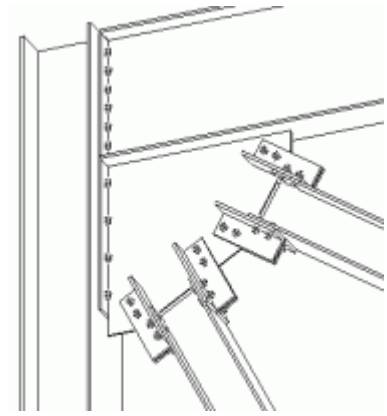
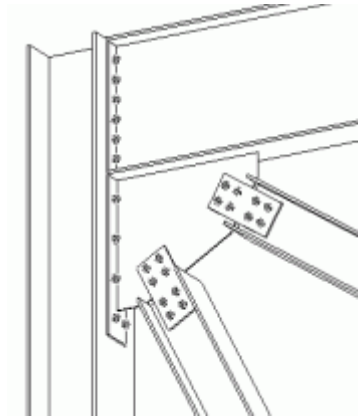
Corner wrapped gusset (63)

Corner wrapped gusset (63) bolts 1 to 10 braces to a gusset plate where two parts form a corner, using clip angles and connection plates. Welds the gusset plate to one of the parts that form the corner.

Objects created

- Gusset plate
- Connection plate between the gusset plate and the brace web
- Shear tab between the gusset plate and the brace flange
- Clip angles
- Shim plates
- Stiffeners
- Bolts
- Welds

Use for

Situation	Description
	Brace profile: W Gusset plate is welded to an extended end plate. Braces are bolted to the gusset plate using clip angles on the flanges and connection plates on the webs.
	Brace profile: W Gusset plate is welded to an extended end plate. Braces are bolted to the gusset plate using clip angles on the flanges.
	Brace profile: W Gusset plate is welded to an extended end plate. Braces are bolted to the gusset plate using connection plates on the webs.

Before you start

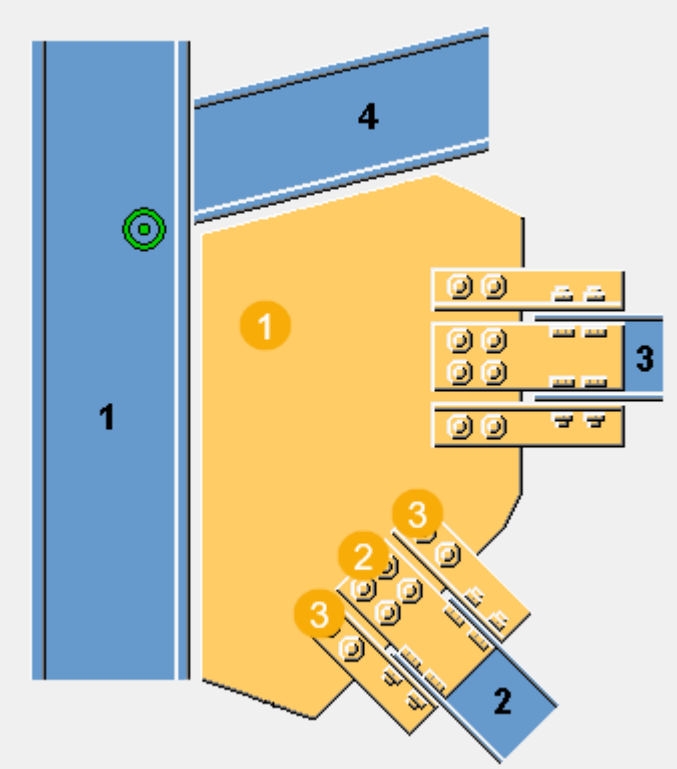
Create two parts that form a corner, and 1 to 10 braces.

NOTE Tekla Structures uses values in the `joints.def` file to create this component.

Selection order

1. Select the main part (the first part that forms the corner).
2. Select the secondary part (first brace).
3. Select the second secondary part (second brace).
4. Select the subsequent secondary parts (subsequent braces).
5. Select the secondary part that forms the corner.
6. Click the middle mouse button to create the component.

Part identification key

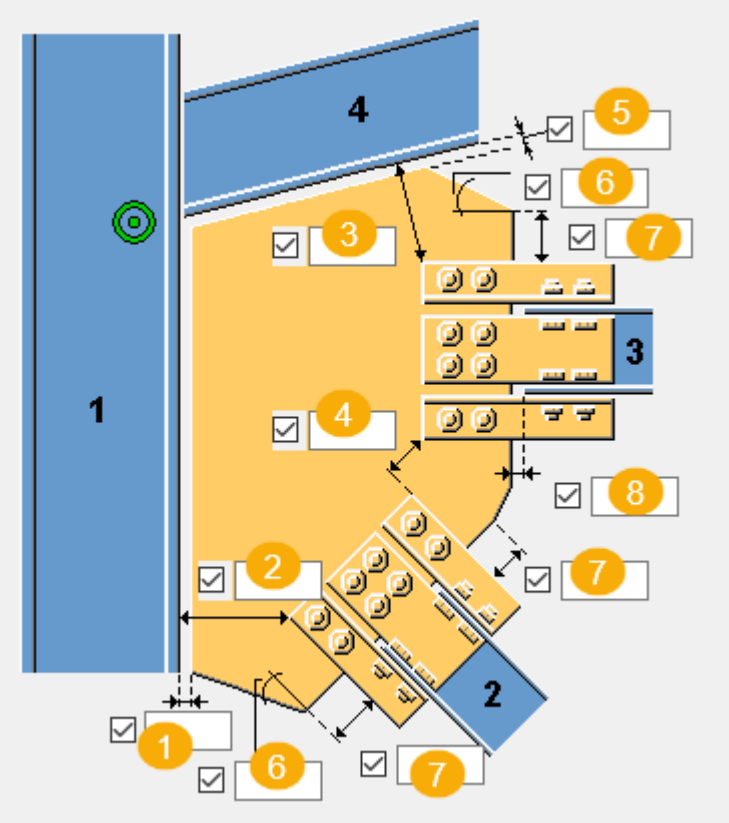


	Description
1	Gusset plate
2	Connection plate
3	Clip angle

Picture tab

Use the **Picture** tab to define the dimensions that control the position and shape of the gusset plate.

Dimensions



	Description
1	Define the gap distance between the gusset plate edge and the main part.
2	Define the edge distance between the main part and the clip angle.
3	Define the edge distance between the clip angle and the last secondary part.
4	Define the edge distance between the clip angles.
5	Define the gap distance between the gusset plate edge and the second part.
6	Define the gusset plate corner angle (in degrees). This value affects the gusset plate shape.
7	Define the length of the edge of the gusset plate. This value affects the gusset plate shape.
8	Define the distance between the gusset plate and the brace.

Gusset tab

Use the **Gusset** tab to control the gusset plate properties.

Gusset plate

Option	Description
Gusset	Thickness, width and height of the gusset plate.

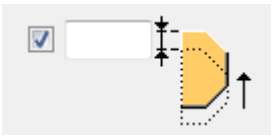
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

NOTE The following examples show only some of the available options. You will find more options on the **Gusset** tab.

Gusset plate position on the brace

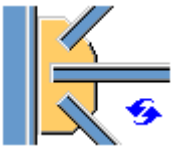
Define where to place the gusset plate on the brace. If needed, you can fine-tune the gusset plate position by moving the gusset plate in the z or in the y direction.

Option	Description
	Default Gusset plate is positioned in the middle of the brace. AutoDefaults can change this option.
	Gusset plate is positioned on the top flange of the brace.
	Define how much the gusset plate is moved in the z direction.

Option	Description
	Define how much the gusset plate is moved in the y direction.

Gusset plate shape

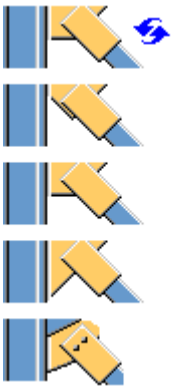
When you select the option to optimize the gusset plate weight, you can define whether the selection order of the braces affects the position of the braces.

Option	Description
	Default AutoDefaults can change this option.
	This option optimizes the gusset plate weight.

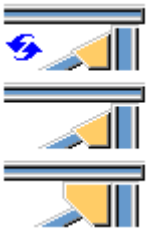
Gusset plate shape

The gusset plate edge can be perpendicular either to the main part or the secondary part.

Option	Description
	Select the gusset plate edge shape between the last and second last secondary part.

Option	Description
	Select the gusset plate egde shape between the main part and the first secondary part.

Gusset plate fitting

Option	Description
	Select whether the gusset is fitted to the last selected secondary part.

Gusset welding

Option	Description
	Default Gusset plate is welded to the main part. AutoDefaults can change this option.
	Gusset plate is welded to the main part.
	Gusset plate is welded to the secondary part.

Brace conn tab

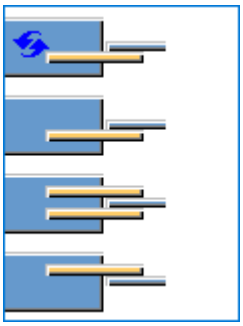
Use the **Brace conn** tab to control connection plate, clip angle, filler plate, and shear tab properties.

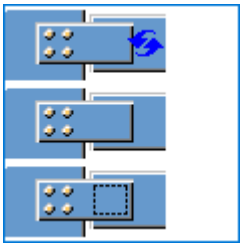
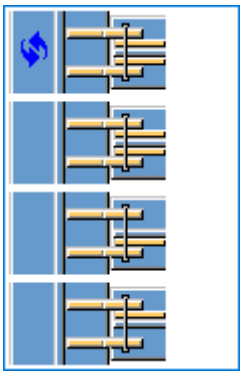
Plates

Option	Description
Connection plate	Thickness and width of the connection plate. Select the connection plate profile.
Upper clip angle **Lower clip angle**	Select the clip angle profile.
Filler plate	Thickness of the filler plate.
Upper shear tab	Thickness, width and height of the upper shear tab.
Lower shear tab	Height of the lower shear tab.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

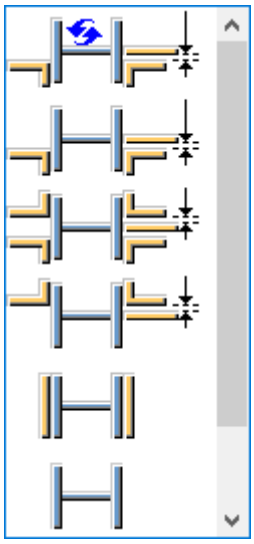
Plate creation

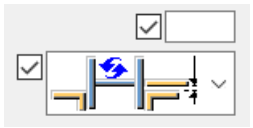
Option	Description
	Select whether one or two connection plates are created.

Option	Description
	Select whether a filler plate is created between the connection plate and the brace web. The default is that a filler plate is not created.
	Select the filler plate creation side. You can use this option when you have selected to create two connection plates.

Clip angle creation

Define whether the braces are attached to the gusset plate using clip angles or shear tabs, and specify the number of clip angles to create. The default option is to create two clip angles below the brace web.



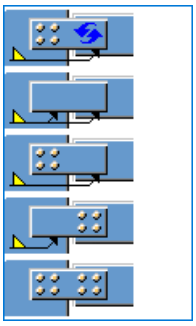
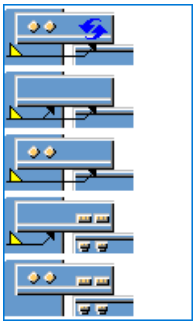


Clip angle orientation

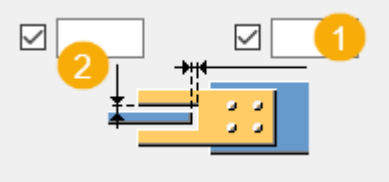
Define how the clip angle is placed on the connection.

Option	Description
	Default Clip angle is placed on the connection so that the longer leg is connected to the gusset plate. AutoDefaults can change this option.
	Clip angle is placed on the connection so that the longer leg is connected to the main part.

Connection type

Option	Description
	Select the connection type (weld or bolts) between the gusset plate and the connection plate.
	Select the connection type (weld or bolt) between the gusset plate and the clip angle.

Connection plate gap dimensions



	Description
1	Horizontal gap dimension
2	Vertical gap dimension

Shim plates tab
Use the **Shim plates** tab to define shim plate properties.

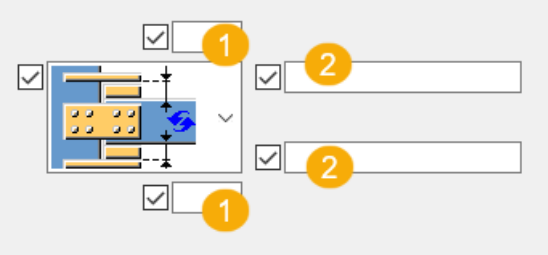
Plates

Option	Description
Shim plate 1 **Shim plate 2** **Shim plate 2**	Thickness, width and height of the shim plate.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Shim plate position

You can create shim plates when connecting braces to the gusset plate using clip angles.



	Description
1	Define the gap between the brace and connection plate.

	Description
2	Define how many shim plates are created at the top and bottom flanges. Enter the shim plate profile numbers: 1, 2 or 3. These are the numbers that are on the upper part on the **Shim plates** tab. For example, if you want to create three shim plates at the top flange, and you want to use **Shim plate 1** twice and **Shim plate 1** once, enter 1 1 2. The first number you enter is the shim plate closest to the brace flange.

Stiffeners tab

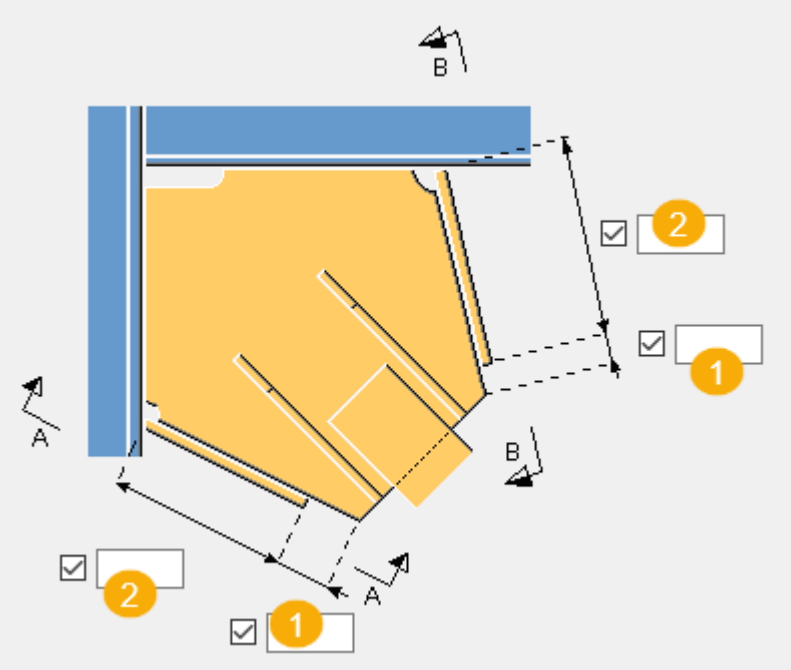
Use the **Stiffeners** tab to control the stiffener properties and dimensions.

Stiffeners

Option	Description
Stiffener 1 **Stiffener 2**	Thickness of the stiffener.

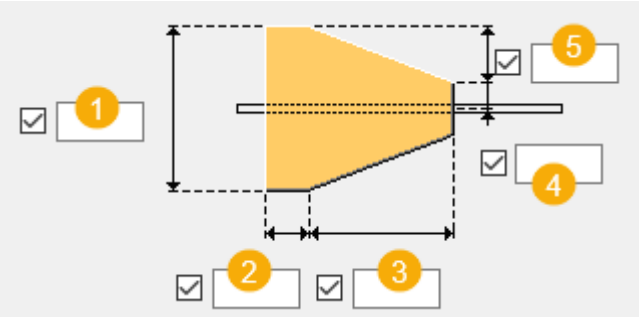
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Stiffener length



	Description
1	Distance between the stiffener edge and the gusset plate edge.
2	Length of stiffener.

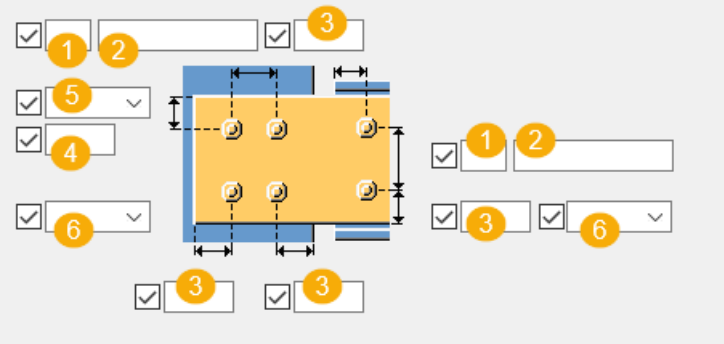
Stiffener dimensions

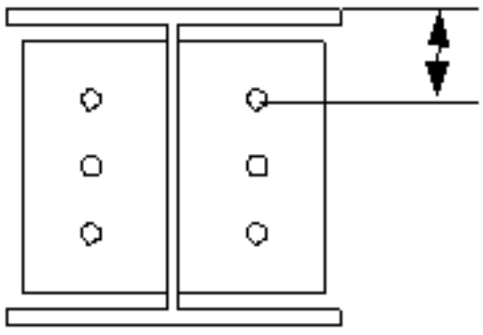


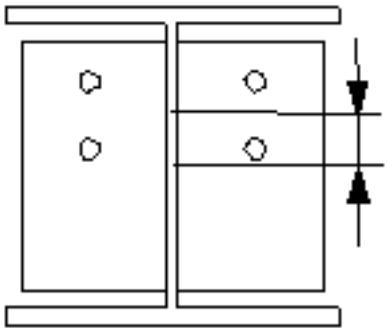
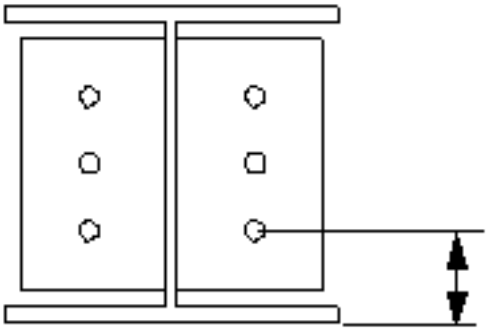
	Description
1	Width of the stiffener.
2	Length of the stiffener base.
3	Length of the skew part of the stiffener.
4	Distance from the stiffener center line.
5	Vertical distance between the stiffener base and the skew part.

Brace bolts 1 / Brace bolts 2 tab
Use the **Brace bolts 1** and **Brace bolts 2** tabs to control the bolts that connect the first and second brace to the gusset plate.

Bolt group dimensions



	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Dimension for vertical bolt group position.
5	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
6	Select the bolt type.

Staggering of bolts

Option	Description
	Default Not staggered AutoDefaults can change this option.
	Not staggered
	Staggered type 1
	Staggered type 2

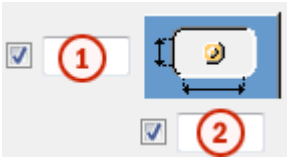
Option	Description
	Staggered type 3
	Staggered type 4

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.



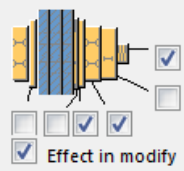
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.

Option	Description	Default
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

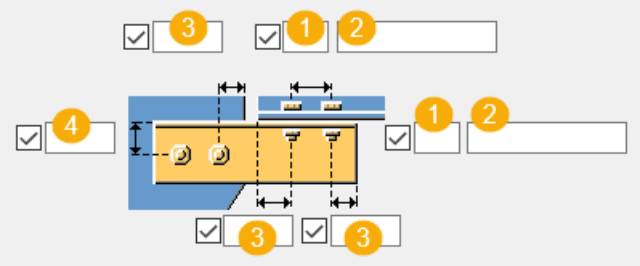
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Angle bolts 1 / Angle bolts 2 tabs

Use the **Angle bolts** tabs to control the bolts that connect the clip angles.

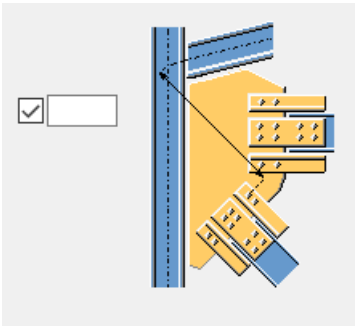
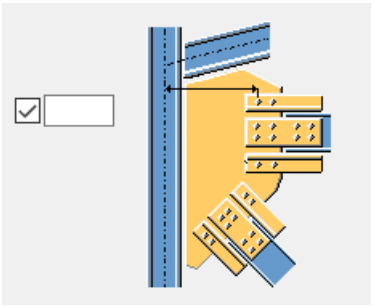
Bolt group dimensions



	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Dimension for vertical bolt group position.

Bolt distance

Define the minimum distance from the connection plate bolts to the intersection point of the main part and brace center lines. If a brace is perpendicular to the main part, the distance is measured from the main part center line to the nearest bolts.

Option	Option
	

Vertical bolt position

Option	Description
	Bolt position from the clip angle edge.

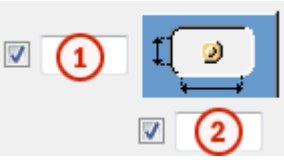
Option	Description
	Bolt position from the secondary part center line.

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`

Slotted holes

You can define slotted, oversized, or tapped holes.



Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	

Option	Description	Default
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Design tab

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

Create welds

Bent gusset (140)

Connects braces in different planes to one or more beams or columns that are in different planes. Creates a gusset plate that is bent along two bending lines. Seals tube or hollow profile braces.

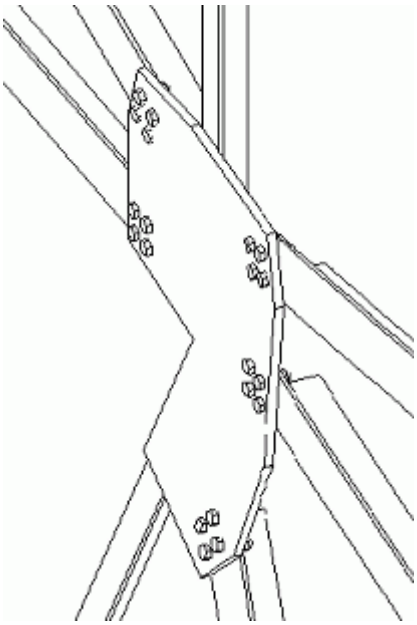
Profiles

Braces: L

Parts created

- Gusset plate
- Seal plates

Use for

Situation	Description
	

Do not use for

Beams or columns that are in the same plane.

Before you start

Create one or more beams and columns, and 1 to 10 braces.

Defining properties

Use the following tabs in the dialog box to define the properties of the parts this component creates:

Tab	Contents	See also
Picture	Dimensions that define gusset plate shape, clearance between braces, cut options for ends of braces.	Modifying gusset plate shape (page 3179)
Gusset	Gusset plate properties, brace notch properties, weld options. Option to cut corner of gusset plate for single brace connections.	Defining gusset plate properties (page 3178) Defining the bend line in skewed connections (140) (page 3182)
Brace conn	Bolt group properties of the bolts connecting the	

Tab	Contents	See also
	gusset plate to the first beam or column picked.	
General	Connection properties, AutoDefaults and AutoConnection rule groups.	
Bracebolts1	Bolt group properties for the bolts that connect the first brace picked to the gusset plate.	
Bracebolts2 **Bracebolts3**	Bolt group properties for the bolts that connect the second and subsequent braces picked to the gusset plate.	
Plates	Options to create filler plates and to bolt braces to both gusset plate and the first or second beam or column picked.	
Analysis	Information used in structural analysis.	

Selection order

1. Beam or column in the first plane
2. First brace
3. Second and subsequent braces in the first plane
4. Braces in planes 2 and 3
5. The beam or column in the second plane
6. Click the middle mouse button to create the component

Heavy brace (165)

Connects a single brace to a gusset plate where 2 parts meet to form a corner.

Profiles

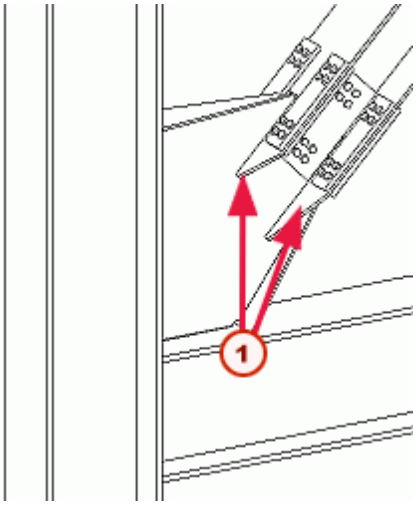
Columns and beams: H, RHS, tube

Braces: W

Parts created

- Gusset plate
- Connection plates
- Rib plates
- Filler plates
- Stiffeners

Use for

Situation	Description
	**Column or beam profile: RHS, tube, I** **Brace profile**: H **Framing type**: Column and beam **Connection method**: • Gusset plate welded column • Brace web bolted to gusset using connection plate. **1** Rib plates

Do not use for

Sloped or skewed beams

Before you start

Create 2 parts that meet to form a corner, and one brace.

Defining properties

Use the following tabs in the dialog box to define the properties of the parts this component creates:

Tab	Contents	See also
Picture	Dimensions of gusset and rib plate, clearance between plate and brace.	
Parts	Properties of stiffeners and gusset, flange, web, rib, flange filler and web filler plates.	Defining plate sizes in marketsizes.dat (165, Japan) (page 3190)

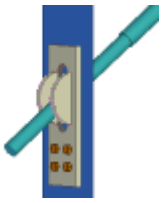
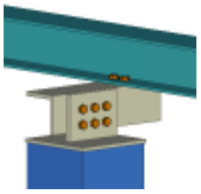
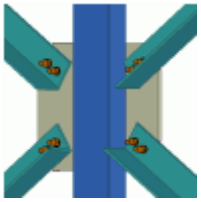
Tab	Contents	See also
Gusset	Gusset plate chamfers, gap between gusset and brace.	
General	Connection properties, AutoDefaults and AutoConnection rule groups.	
Ribs	Dimensions that define the size and location of rib plates, option to create ribs of equal length.	Defining rib plate dimensions (165) (page 3191)
Stiffeners	Dimensions of stiffener plates.	
Web bolts	Bolt group properties for the bolts fastening the web connection plate to the gusset plate.	
Flange bolts	Bolt group properties for the bolts fastening the flange connection plate to the gusset plate.	
Analysis	Information used in structural analysis.	

Selection order

1. First part that forms the corner
2. First brace
3. Second and subsequent braces
4. Second part that forms the corner
5. Click the middle mouse button to create the component

Windbracing connections

Use the following components to automatically create windbracing:

Component	Image	Description
Windbracing (1) (page 3147)		Connects a single brace to a beam or column, using a connection plate, round plates, a threaded rod, and nuts.
Wind column (5) (page 3149)		Connects a rafter to a wind column by using a single L profile or a twin seat profile.
Windbracing 2 (16) (page 3157)		Connects a single hollow brace to a beam or a column, using a connection plate, threaded rod, and nuts.
Windbrace connection (110) (page 3172)		Bolts braces to a gusset plate. Welds or bolts the gusset plate to the main part. Braces can be: • Located in the same work plane as the beam or column, or skewed • On the same side of the beam or column, or on each side

Windbracing (1)

Connects a single hollow brace to a beam or column, using a connection plate, threaded rod, and nuts. Welds round plates to the connection plate and bolts it to the beam or column.

WARNING Tekla Structures creates the threaded rod using the **Detailing** --> **Attach Part** command. This means that the rod and brace are treated as a single part when you use commands like **Delete**, **Move** or **Copy**.

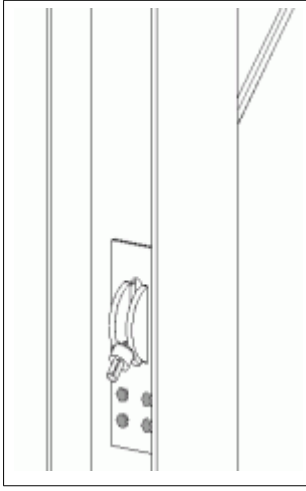
Profiles

Brace: RHS, tube

Parts created

- Connection plate
- Round plates (2)
- Filler plate
- Threaded rod
- Nuts on rod (2)

Use for

Situation	Description
	**Column or beam profile: H** **Brace profile**: RHS

Before you start

Create a beam or column and one brace.

Defining properties

Use the following tabs in the dialog box to define the properties of the parts this component creates:

Tab	Contents	See also
Picture	Dimensions that define the size and position of the connection and round plates.	
Parts	Properties of all plates, nuts, and washers.	Defining part thickness (1) (page 3193)
Brace conn	Properties of the hole in the beam or column and the slot in the connection plate, position of round plates, rod dimensions.	Defining hole type (1) (page 3193)

Tab	Contents	See also
General	Connection properties, AutoDefaults and AutoConnection rule groups.	
Bolts	Bolt group properties for the bolts that fasten the connection plate to the beam or column.	
Analysis	Information used in structural analysis.	

Selection order

1. Beam or column
2. Brace

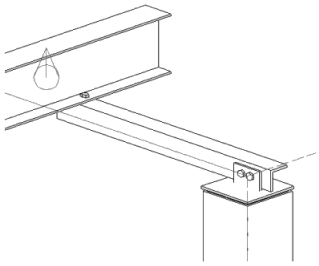
Wind column (5)

Wind column (5) connects a rafter to a wind column by using a single L profile or a twin seat profile. A cap plate is added to the wind column, and a vertical plate on top of the cap plate to be connected to the seat. The seat can be created above or below the rafter, and the wind column is adjusted accordingly. The seat can either be bolted or welded to the rafter and the vertical plate.

Objects created

- Vertical plate
- Cap plate
- L/T seat profile
- Bolts
- Welds

Use for

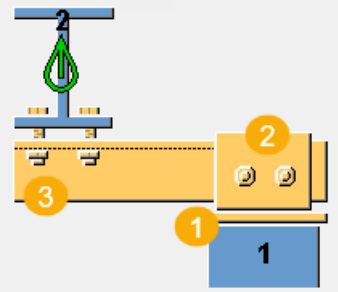
Situation	Description
	Rafter connected to a wind column.

Selection order

1. Select the main part (wind column).
2. Select the secondary part (rafter).

   The connection is created automatically when the secondary part is selected.

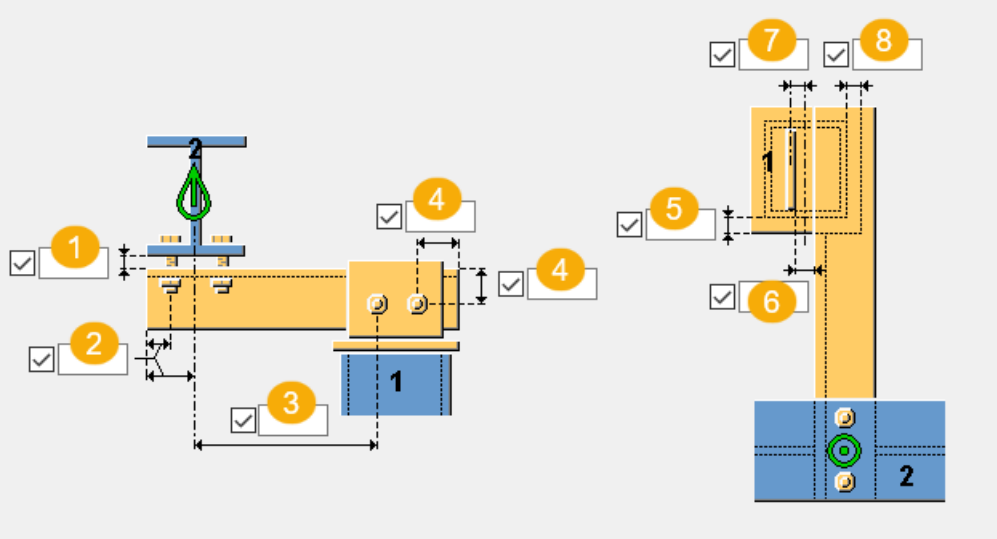
Part identification key



	Description
1	Cap plate
2	Vertical plate
3	L/T seat profile

Picture tab
Use the **Picture** tab to define the connection dimensions.

Dimensions



	Description
1	Distance between the rafter and the seat.

	Description
2	If the seat is bolted, this dimension is the bolt edge distance of the seat on the rafter side. If the seat is welded, this dimension is the distance from the rafter center line to the edge of the seat.
3	Distance from the rafter center to the nearest plate bolt.
4	Bolt edge distance.
5	Cap plate edge extension in the direction parallel to the seat.
6	Distance between the vertical plate and the seat.
7	Distance between the column center line and the vertical plate.
8	Cap plate edge extension in the direction perpendicular to the seat.

Parts tab

Use the **Parts** tab to define the part properties.

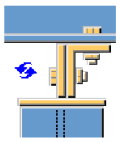
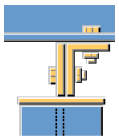
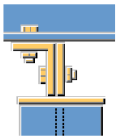
Parts

Option	Description
Vertical plate	Thickness of the vertical plate.
Cap plate	Thickness of the cap plate.
L/T profile	Select the profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Parameters tab
Use the **Parameters** tab to define the seat type, orientation, and position.

Seat type

Option	Description
	Default Seat at the near side. AutoDefaults can change this option.
	Seat at the near side.
	Seat at the far side.
	Seat at both sides.

Seat orientation

Option	Description
	Default Longer leg is horizontal. AutoDefaults can change this option.
	Longer leg is horizontal.
	Longer leg is vertical.

Seat position

Option	Description
	Default Seat is below the rafter. AutoDefaults can change this option.

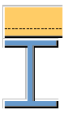
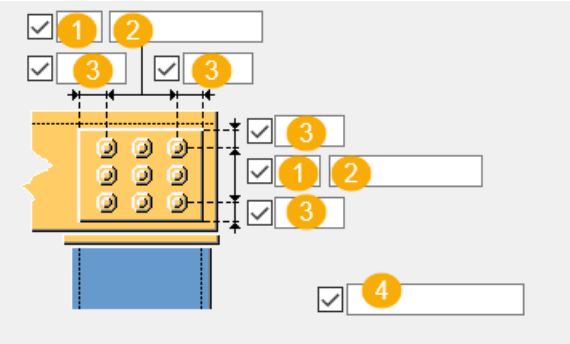
Option	Description
	Seat is below the rafter.
	Seat is above the rafter.

Plate bolts tab

Use the **Plate bolts** tab to define the bolt group dimensions, bolt properties, and the vertical plate connection to the seat.

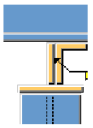
Bolt group dimensions



	Description
1	Number of bolts.
2	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Define which bolts are deleted from the bolt group. Enter the bolt numbers of the bolts to be deleted and separate the numbers with a space. Bolt numbers run from left to right and from top to bottom.

Connection type

Select how the vertical plate is connected to the seat.

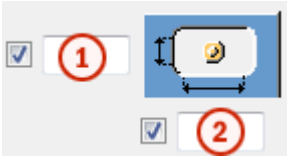
Option	Description
	Default Bolted AutoDefaults can change this option.
	Bolted
	Welded

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

Slotted holes

You can define slotted, oversized, or tapped holes.

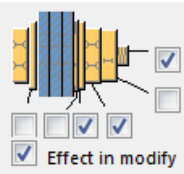


Option	Description	Default
1	Vertical dimension of slotted hole.	0, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	0, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

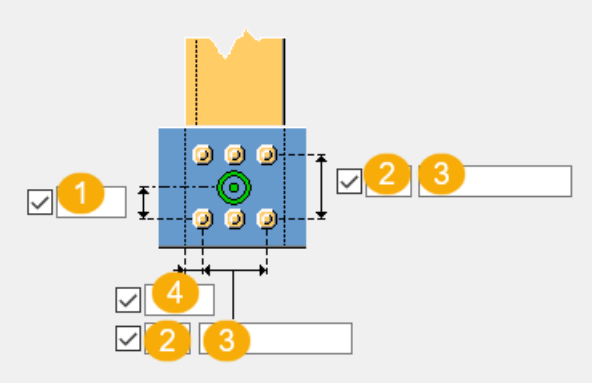
Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

Beam bolts

Use the **Beam bolts** tab to define the bolt group dimensions, bolt properties, and the rafter connection to the seat.

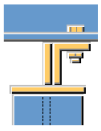
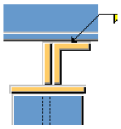
Bolt group dimensions



	Description
1	Distance between the rafter center and the bolts.
2	Number of bolts.
3	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
4	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.

Connection type

Select how the rafter is connected to the seat.

Option	Description
	Default Bolted AutoDefaults can change this option.
	Bolted
	Welded

Bolt basic properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog.
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog.
Tolerance	Gap between the bolt and the hole.	
Thread in mat	Defines whether the thread may be within the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	`Yes`
Site/Workshop	Location where the bolts should be attached.	`Site`

General tab

Click the link below to find out more:

General tab

Design tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

Welds

Click the link below to find out more:

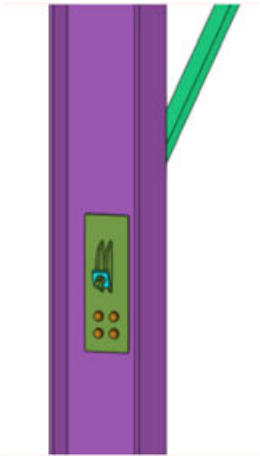
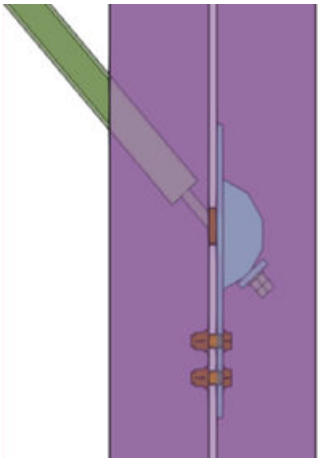
Windbracing 2 (16)

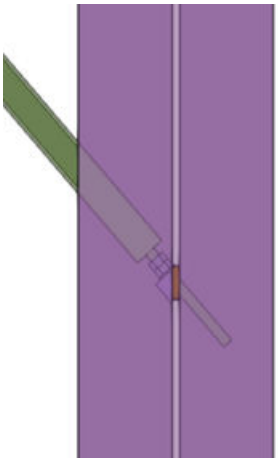
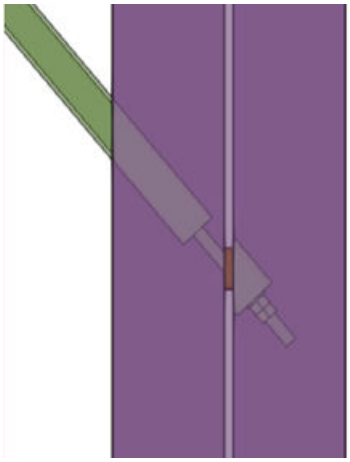
Windbracing 2 (16) connects a single hollow brace to a beam or a column, using a connection plate, threaded rod, and nuts. Round plates are welded to the connection plate and the connection plate bolted to the beam or column.

Objects created

- Connection plate
- Round plates
- Shim plate
- Threaded rod
- Nuts on rod
- Tube profile

Use for

Situation	Description
	Windbracing, connection plate is bolted to the column. Round plates are welded to the connection plate.
	Windbracing, connection plate is bolted to the column. Round plates are welded to the connection plate.

Situation	Description
	Windbracing, without round plate.
	Windbracing, triangular plate.

Before you start

Create a beam or a column and one brace.

Selection order

1. Select the beam or the column (main part).
2. Select the brace (secondary part).

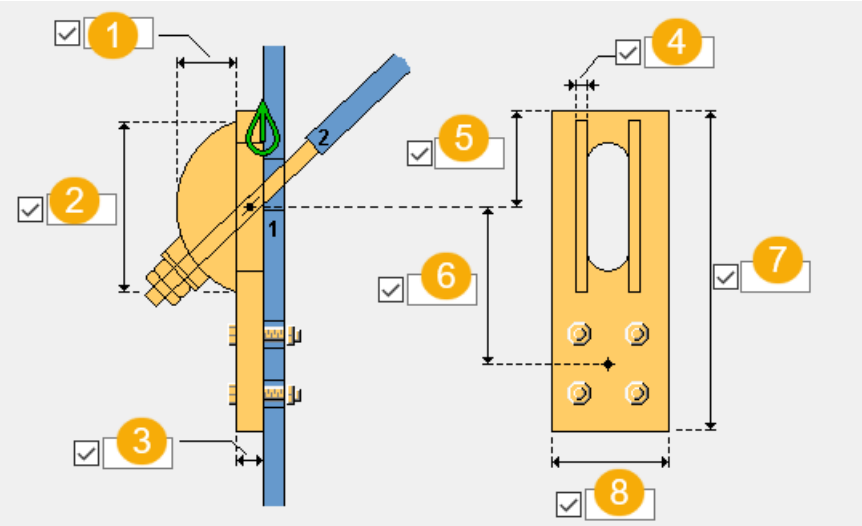
   The connection is created automatically when you select the secondary part.

Part identification key

	Description
1	Beam - main part
2	Brace - secondary part
3	Windbracing connection with connection plate, round plate, and bolts

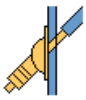
Picture tab
Use the **Picture** tab to define the size and the position of the connection plate and the round plate.

Plate dimensions



	Description
1	Width of the round plate.
2	Height of the round plate.
3	Thickness of the connection plate.
4	Thickness of the round plate.
5	Distance from the top of the connection plate to the center of the slotted hole.
6	Vertical distance from the center of the slotted hole to the center of the bolt group.
7	Height of the connection plate.
8	Width of the connection plate.

Round plate or tube

Option	Description
	Round plates with a threaded rod.
	Tapered tube with a threaded rod.
	Threaded rod.
	Triangular plate on the tie rod side with a threaded rod.
	Triangular plate at the end side with a threaded rod.
	Triangular plates on both sides with a threaded rod.
	No plates or a threaded rod.
	Triangular plate on the tie rod side.
	Triangular plate at the end side.

Option	Description
	Triangular plates on both sides.
	No plates or a threaded rod.
	Rod welded to an L profile.

Create assembly with connecting parts

Define whether or not the connecting parts (shim plate, washer, nuts) are welded together.

Weld connecting parts

Define which parts are welded.

Parts tab

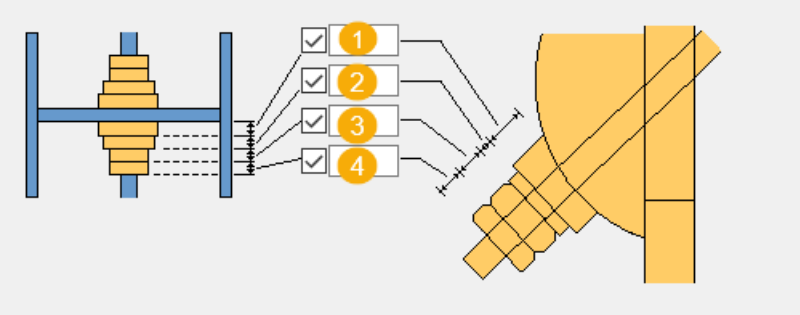
Use the **Parts** tab to define the part properties.

Parts

Option	Description
Connection plate	Thickness, width, and height of the connection plate.
Round plate	Thickness, width, and height of the round plate.
Filler	Select the profile from the profile catalog. Tekla Structures creates a simplified shim plate. In reality, the plate is curved on one side, equal to the radius of the round plate. Enter the width and height, for example `PL50*50` or `50*50`.
Washer	Select the profile from the profile catalog.
Nut	Select the profile from the profile catalog.
Extra nut	Select the profile from the profile catalog.
Tube	Select the profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Comment	Add a comment about the part.	

Thickness dimensions



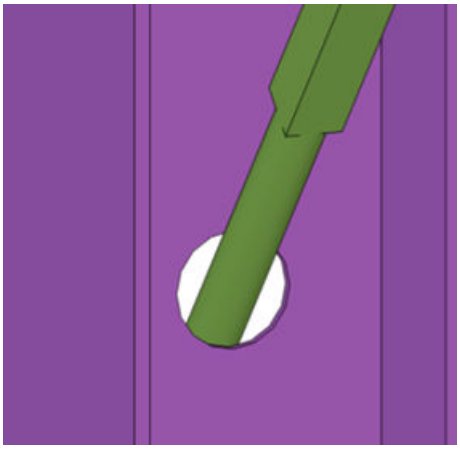
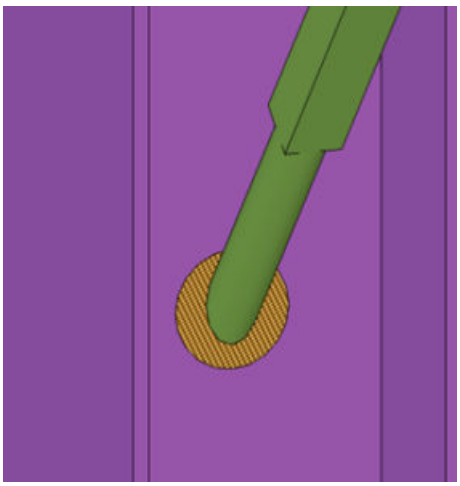
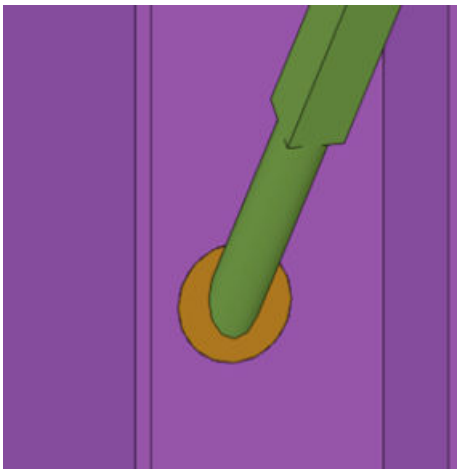
	Description
1	Thickness of the shim plate.
2	Thickness of the washer.
3	Thickness of the nut.
4	Thickness of the extra nut.

Parameters tab

Use the **Parameters** tab to define the rod and hole dimensions.

Hole creation

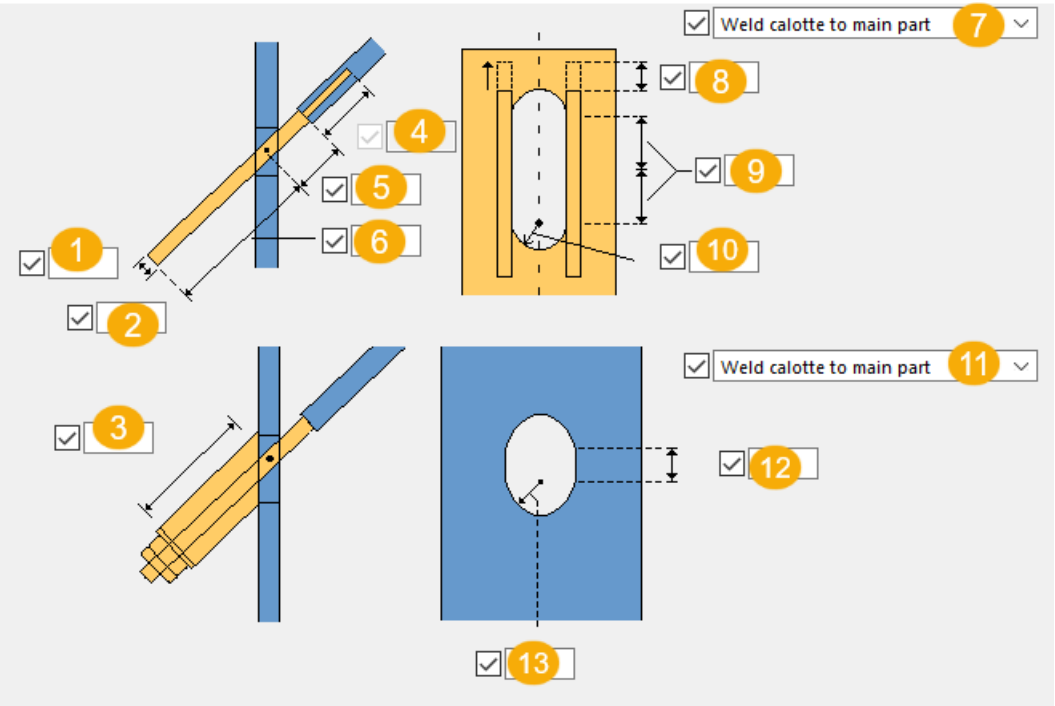
Define how the hole is created in the main part.

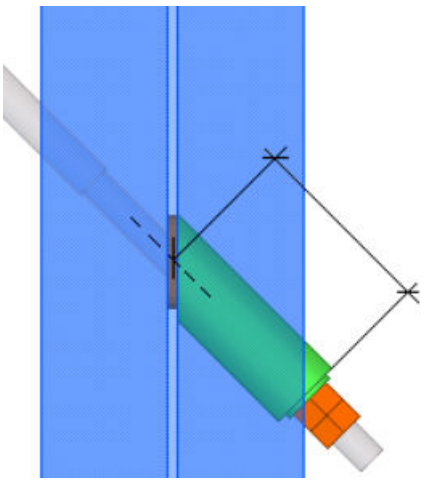
Option	Example
Hole by part cut	
Hole by bolt	
Hole by part cut and bolt	

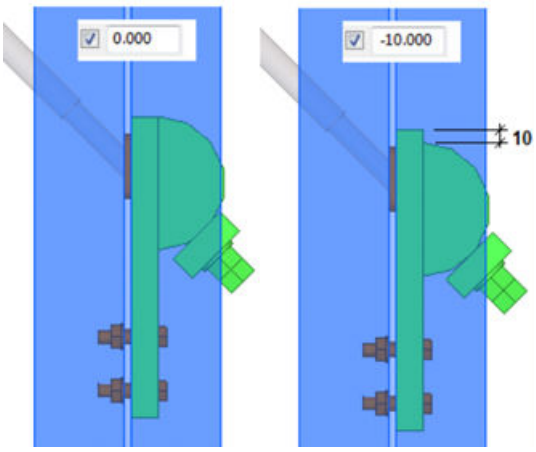
Connection method

Define how the round plates are joined to the connection plate.

Dimensions



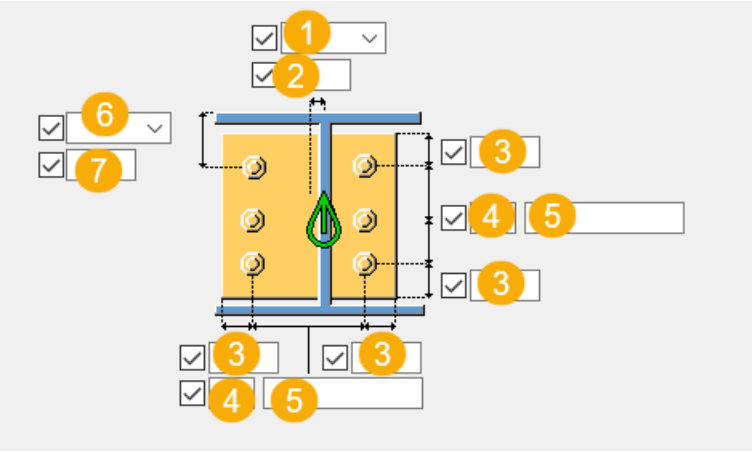
	Description
1	Diameter of the rod end.
2	Offset from the secondary part edges.
3	Length of the connecting piece. You can define this if a tube is created. Length is measured along the center line. 
4	Slot length in the main part.
5	Rod length to the center of the main part.
6	Rod length to the center of the main part.

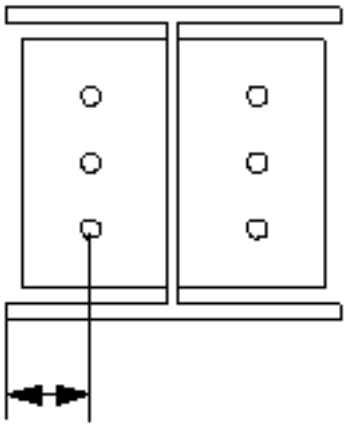
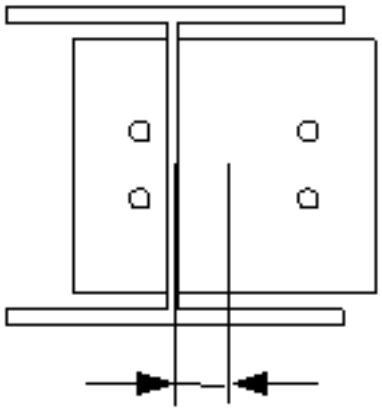
	Description
7	Define how the callote is welded. When not welded, the calotte is connected to the main part with a bolted connection.
8	Vertical offset of the round plates. Define a value to offset the plates in the vertical direction. 
9	Slotted hole height.
10	Slotted hole width.
11	Define how the rod connection is welded.
12	Slotted hole height in the main part.
13	Slotted hole width in the main part.

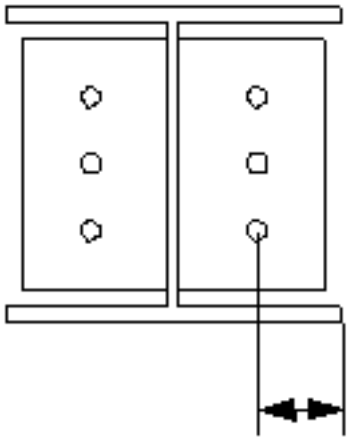
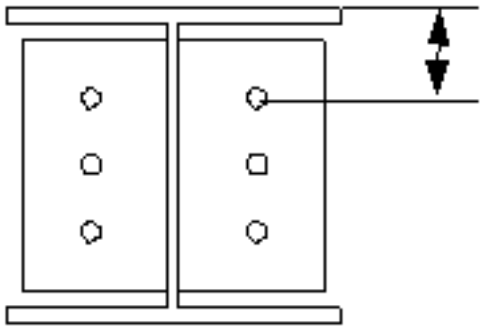
Bolts tab

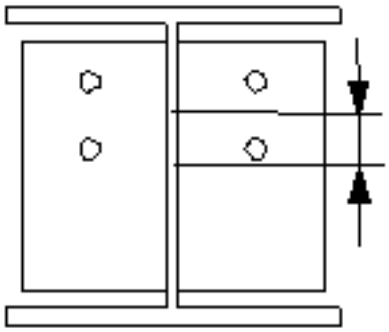
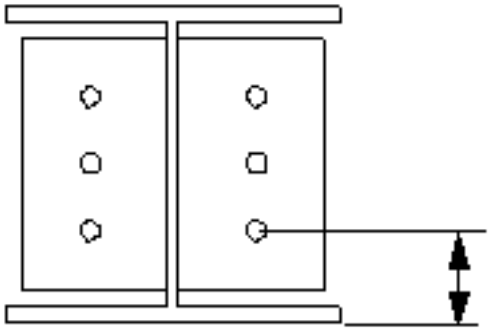
Use the **Bolts** tab to define the bolt group dimensions and bolt properties.

Bolt group dimensions



	Description
1	Select how to measure the dimensions for horizontal bolt group position. • **Left**: From the left edge of the secondary part to the leftmost bolt.  • **Middle**: From the center line of the secondary part to the center line of the bolts. 

	Description
	• **Right**: From the right edge of the secondary part to the rightmost bolt. 
2	Dimension for horizontal bolt group position.
3	Bolt edge distance. Edge distance is the distance from the center of a bolt to the edge of the part.
4	Number of bolts.
5	Bolt spacing. Use a space to separate bolt spacing values. Enter a value for each space between bolts. For example, if there are 3 bolts, enter 2 values.
	Select how to measure the dimensions for vertical bolt group position. • **Top**: From the upper edge of the secondary part to the uppermost bolt. 

	Description
	• **Middle**: From the center line of the bolts to the center line of the secondary part.  • **Below**: From the lower edge of the secondary part to the lowest bolt. 
	Dimension for vertical bolt group position.

Basic bolt properties

Option	Description	Default
Bolt size	Bolt diameter.	Available sizes are defined in the bolt assembly catalog. `16 mm`
Bolt standard	Bolt standard to be used inside the component.	Available standards are defined in the bolt assembly catalog. `4014-8.8`
Tolerance	Gap between the bolt and the hole.	`4 mm`
Thread in mat	Defines whether the thread may be within	`Yes`

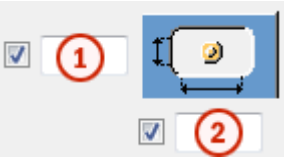
Option	Description	Default
	the bolted parts when bolts are used with a shaft. This has no effect when full-threaded bolts are used.	
Site/Workshop	Location where the bolts should be attached.	`Site`

Cut length

Defines the depth at which Tekla Structures searches for the sections of the bolted parts. You can determine whether the bolt will go through one flange or two.

Slotted holes

You can define slotted, oversized, or tapped holes.



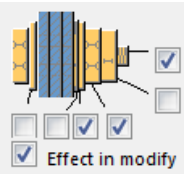
Option	Description	Default
1	Vertical dimension of slotted hole.	`0`, which results in a round hole.
2	Horizontal dimension of slotted hole, or allowance for oversized holes.	`0`, which results in a round hole.
Hole type	**Slotted** creates slotted holes. **Oversized** creates oversized or tapped holes. **No hole** does not create holes.	
Rotate Slots	When the hole type is **Slotted**, this option rotates the slotted holes.	
Slots in	Part(s) in which slotted holes are created. The options depend on the component in question.	

Examples of slotted holes

Bolt assembly

The selected check boxes define which component objects (bolt, washers, and nuts) are used in the bolt assembly.

If you want to create a hole only, clear all the check boxes.



To modify the bolt assembly in an existing component, select the **Effect in modify** check box and click **Modify**.

Bolt length increase

Define how much the bolt length is increased. Use this option when, for example, painting requires the bolt length to be increased.

UDA tab

Use the **UDA** tab to define the user-defined attributes for the connection plate.

You can define the fabricator name, type, nomination, article number, product code, and product description.

General

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Welds

Click the link below to find out more:

Windbrace connection (110)

Connects 1 to 10 braces to a beam or column by bolting them to a gusset plate, and welding or bolting the gusset plate to the beam or column. The braces can be:

- Located in the same work plane as the beam or column, or skewed
- On the same side of the beam or column, or on each side

Profiles

Braces: L

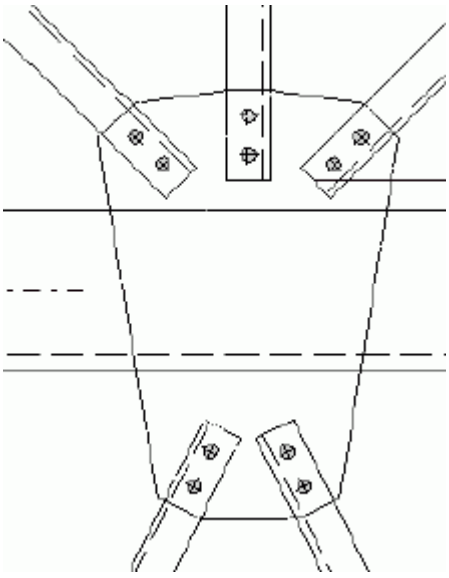
Parts created

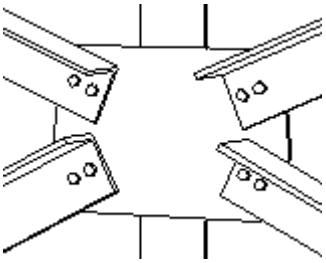
- One gusset plate, if the braces are located on the same side of the beam or column

OR

- One gusset plate on each side of the beam or column, if the braces are located on either side of the beam or column. The plates are welded in the shop to create a folded gusset plate.

Use for

Situation	Description
	Two gusset plates (welded in the shop to form a folded plate) **Brace profile**: L **Connection method**: Braces bolted directly to gusset plate

Situation	Description
	**Column or beam profile: L** **Brace profile**: L **Connection method**: • Gusset plate welded column • Braces web bolted to gusset plate. No stiffeners.

Before you start

Create a beam or column and 1 to 10 braces.

Defining properties

Use the following tabs in the dialog box to define the properties of the parts this component creates:

Tab	Contents	See also
Picture	Dimensions that define gusset plate shape; brace clearances and cut options for ends of braces.	Modifying gusset plate shape (page 3179)
Gusset	Gusset plate properties, brace notch properties, weld options.	Modifying gusset plate shape (page 3179) Defining gusset plate type (page 3180) Defining gusset plate position on the brace (page 3182) Creating tongue plates (20, 22, 56) (page 3185) Creating an orientation hole in the gusset plate (110) (page 3183) Dimensioning holes in drawings (110, 140) (page 3194)
Gusset conn	Bolt group properties of the bolts connecting the gusset plate to the beam or column.	
General	Connection properties, AutoDefaults and	

Tab	Contents	See also
	AutoConnection rule groups.	
Bracebolts1	Bolt group properties for the bolts that connect the braces to the upper, or only gusset plate.	
Bracebolts2	Bolt group properties for the bolts that connect the braces to the lower gusset plate.	
Brace conn	Option to create filler plates between the braces and the gusset plate; option to bolt the braces to the beam or column, as well as the gusset plate.	Double-bolting braces (110) (page 3187)
Analysis	Information used in structural analysis.	

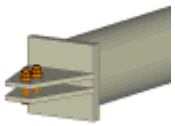
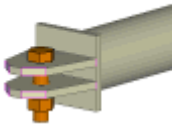
Selection order

1. Beam or column
2. First brace
3. Second and subsequent braces
4. Click the middle mouse button to create the component

Bracing connection elements

Use the following components to create individual bracing elements, such as stiffeners and gusset plates. You can then use these elements to connect bracing to one or more parts, or in custom components.

Component	Image	Description
Crushed tube in points (S46) (page 3175)		Creates a tube brace between two points using a tube crushed between bolts.

Component	Image	Description
Gusset tube in points (S47) (page 3176)		Creates a tube brace between two points that you pick, using a circular hollow section with 2 Tees at each end.
Crushed tube in bolts (S48) (page 3177)		Creates a tube brace using a tube crushed between two existing bolts.
Gusset tube in bolts (S49) (page 3177)		Creates a tube brace between two bolts, with tees at both ends.

Crushed tube in points (S46)

Creates a brace between two points that you pick, using a circular hollow section. Creates bolt groups at each end of the brace.

WARNING You cannot modify the individual objects that this component creates.

Parts created

Tube brace

Before you start

Create 2 points.

WARNING To adjust the rotation of the tees, change the work plane before you create the connection.

Defining properties

Use the following tabs in the dialog box to define the properties of the parts this component creates:

Tab	Contents
Picture	Dimensions that define the location of the bolts.
Parts	Properties of the tube brace.

Tab	Contents
Tube pos.	Options that affect the position, plane and class of the tube.
Bolts	Bolt group properties.

Selection order

1. Start point of the tube
2. End point of the tube
3. Click the middle mouse button to create the component

Gusset tube in points (S47)

Creates a brace between two points that you pick, using a circular hollow section with 2 Tees at each end. Seals tube.

WARNING You cannot modify the individual objects that this component creates.

Parts created

- Tube
- Tees (4)
- End plate (2)

Before you start

Create two points. See .

WARNING To adjust the rotation of the tees, change the work plane before you create the connection.

Defining properties

Use the following tabs in the dialog box to define the properties of the parts this component creates:

Tab	Contents
Picture	
Parts	Properties of the tube, end plate, and tees
Tube pos.	Options that affect the position, plane and class of the tube.
Bolts	Bolt group properties.

Selection order

1. Start point of the tube

2. End point of the tube
3. Click the middle mouse button to create the component

Crushed tube in bolts (S48)

Creates a brace using a crushed tube between two existing bolts.

WARNING You cannot modify the individual objects that this component creates.

Parts created

Tube

Before you start

Create two bolts, either manually or using a detailing component (Standard gusset (1065) (page 2515)).

Defining properties

Use the following tabs in the dialog box to define the properties of the parts this component creates:

Tab	Contents
Picture	Dimensions that define the crushed end of the tube.
Parts	Properties of the tube.
Parameters	Options that affect the position, plane and class of the tube.
Bolts	

Selection order

1. First bolt
2. Second bolt
3. Click the middle mouse button to create the component

Gusset tube in bolts (S49)

Creates a brace using a tube between two bolts, with a built-up T profile at both ends. Seals tube.

WARNING You cannot modify the individual objects that this component creates.

Parts created

- Tube

- Tees (2)
- End plate

Before you start

Create 2 bolts on the plane where you want to create the brace.

Defining properties

Use the following tabs in the dialog box to define the properties of the parts this component creates:

Tab	Contents	See also
Picture	Dimensions that define the end of the tube and the tees.	
Parts	Properties of the tube, end plate and Tees.	
Parameters	General connection properties.	
Bolts	Properties of bolts.	

Selection order

1. First bolt
2. Second bolt
3. Click the middle mouse button to create the component

Defining gusset plate properties

This section explains how to define gusset plate properties for various bracing components.

Click the links below to find out more:

- Modifying gusset plate shape (page 3179)
- Defining gusset plate type (page 3180)
- Selecting gusset plate connection material (11) (page 3180)
- Defining wraparound gusset plates (58, 59, 60) (page 3180)
- Defining gusset plate location (67) (page 3181)
- Defining gusset plate position on the brace (page 3182)
- Defining gusset plate position on the beam or column (11) (page 3182)
- Defining the bend line in skewed connections (140) (page 3182)
- Specifying gusset plate connection material (11, 20, 62) (page 3183)

- Creating an orientation hole in the gusset plate (110) (page 3183)

NOTE Not all component dialog boxes contain all the properties listed in this section.

Modifying gusset plate shape

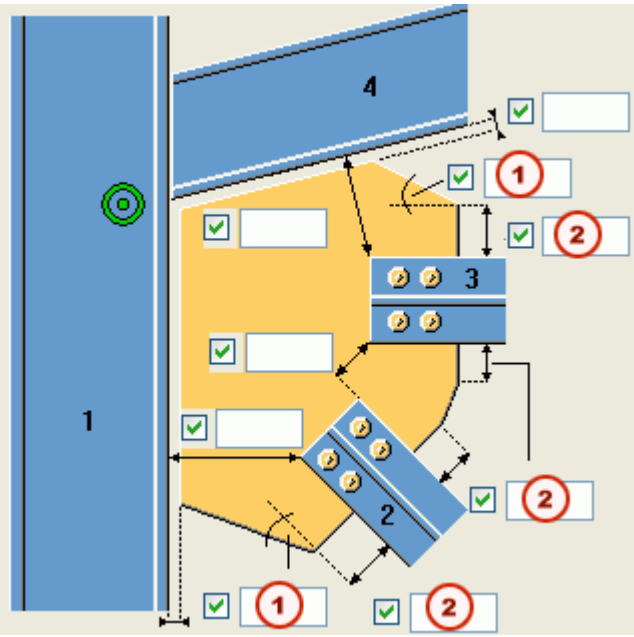
To change shape of a gusset plate, you need to change its dimensions. Bracing components automatically create gusset plates based on the dimensions in the `joints.def` file.

- To only modify the dimensions of the gusset plate for the connections you have selected, enter the new dimensions on the **Picture** tab.
- To modify the dimensions of the gusset plate for all connections that this component creates, edit the dimensions in the `joints.def` file. Tekla Structures uses values in the `joints.def` file to create this component. See Define connection properties in the joints.def file.

Using the dialog box

To modify the shape of the gusset plate using the dialog box:

1. Go to the **Picture** tab and enter values in any of the following fields (this image is from **Corner bolted gusset (57)**):



1	Enter angles here.
2	Enter dimensions here.

2. Click the **Apply** button to change any connections you have open.

Defining gusset plate type

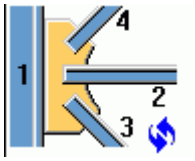
To define the type of gusset plate to use, go to the **Gusset** tab. The default option is:

Optimizing plate weight

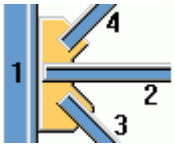
To optimize plate weight, select this option:

Picking order affects brace position

Optional. When you select the option to optimize plate weight, you can also choose to have the picking order affect the position of the braces. The default option is 'No':



To place the first brace picked closest to the first beam or column picked, select this option:



Selecting gusset plate connection material (11)

To select the connection material to use to connect the gusset plate to the beam or column, go to the **Gusset** tab. The default option is to weld the gusset plate directly to the beam or column:

There are also options to use clip angles or shear tabs.

Defining wraparound gusset plates (58, 59, 60)

If the gusset plate clashes with the column, Tekla Structures notches the gusset plate to wrap around the column. To specify the angle of the notch, go to the **Gusset** tab. The default is:

Notch chamfer type

To specify the chamfer type to use for the notch, go to the **Gusset** tab. The options are:

Option	Description
	No chamfer. Notch edges parallel to the edges of the gusset plate.
	Default.
	Creates a rounded chamfer calculated as a quarter of a circle. Enter the diameter of the circle as the horizontal dimension of the chamfer. See Defining chamfer dimensions.
	Creates a rounded chamfer calculated as a three-quarters of a circle. Enter the diameter of the circle as the horizontal dimension of the chamfer. See Defining chamfer dimensions. The center point of the circle is the corner of the notch.
	Bevels the corner of the gusset plate.
	Does not notch or chamfer the corner of the gusset plate.

Defining chamfer dimensions

To define the size of the notch chamfer, or the diameter of the circle used to calculate rounded chamfers, go to the **Gusset** tab and enter these dimensions:

Defining gusset plate location (67)

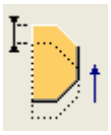
To specify the location of the gusset plate relative to the beam or column web, go to the **Gusset** tab. The default location is above the beam or column web:

Defining gusset plate position on the brace

To indicate where to locate the gusset plate on the brace, go to the **Gusset** tab. The default option is 'Middle':

Fine-tuning position

To fine-tune the position of a gusset plate, enter values for the following properties:

Property	Description
	Moves the gusset plate in the y direction.
	Moves the gusset plate towards the connection.

Defining gusset plate position on the beam or column (11)

To indicate where to locate the gusset plate on the beam or column, go to the **Gusset** tab. The default option is parallel to the brace:

Defining the bend line in skewed connections (140)

To define the bend line in skewed connections, enter the following dimensions on the **Gusset** tab.

Specifying gusset plate connection material (11, 20, 62)

To specify what type of connection material to use to connect the gusset plate to the beam or column, go to the **Gusset** tab. The default option is to connect the gusset plate to directly to the flange of the beam or column:

You can also choose to use clip angles or fastener plates in various configurations.

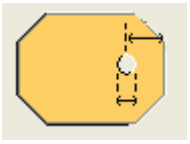
Creating an orientation hole in the gusset plate (110)

You may need to create an orientation hole in the gusset plate to indicate the position of the plate when the connection is assembled in the shop, or during erection. To create an orientation hole in the gusset plate, go to the **Gusset** tab:

The default option is no orientation hole:

To create the hole in the left or right of the gusset plate:

- Select one of the options in the listbox
- Enter the dimensions of the hole:



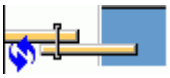
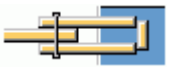
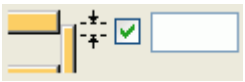
Defining brace connection properties

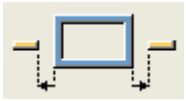
Click the links below to find out more:

- Defining the brace connection (page 3184)
- Creating tongue plates (20, 22, 56) (page 3185)
- Notching brace flange (11, 57) (page 3185)
- Notching brace flange (60) (page 3186)
- Notching the brace (22, 59) (page 3186)
- Cutting braces (60) (page 3187)
- Double-bolting braces (110) (page 3187)
- Using clip angles to connect braces (11, 57) (page 3187)
- Creating clip angles or shear tabs (58, 61, 62, 63) (page 3188)

Defining the brace connection

Several component dialog boxes contain a **Brace conn** tab, where you can specify the options to use to connect the brace to the gusset plate. Some of the options are:

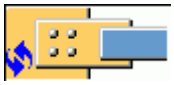
Option	Description
How many connection plates to use.	You can use one or two connection plates to connect the brace to the gusset plate. The default option is 1 plate:  If there are two connection plates, Tekla Structures creates an end plate to the ends of the connection plates.  You can specify the width of the end plate. 
Notch the brace or cut the connection plate?	The default option is to notch the brace:

Option	Description
	If you choose to cut the plate, you can specify the gap between the brace and the plate: 
How to connect the brace to the connection plate.	The default option is to weld the brace: You can also bolt the brace, or weld it and notch it around the nuts connecting it to the gusset plate:

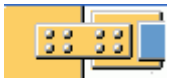
Creating tongue plates (20, 22, 56)

To connect the brace to the gusset plate using a tongue plate and cover plate, go the **Brace conn** tab.

By default, a tongue plate is not created:



To create a tongue plate, select this option:



Notching brace flange (11, 57)

Some components give you the option to notch the brace flange. You may want to do this:

- if the gusset plate collides with the brace flange
- to create slots in hollow braces. See Creating slots in hollow braces (11, 57)
- to connect hollow braces using clip angles. See Using clip angles to connect braces (11, 57) (page 3187)

To notch the flange or create a slot, go to the **Gusset** tab. The default option is not to notch:

Creating slots in hollow braces (11, 57)

To create a slot in a hollow brace, go to the **Gusset** tab:

1. Define the position of the gusset plate as 'Middle'. This is the default option. See Defining gusset plate position on the brace (page 3182).
2. Select this option to create a slot in the brace and fasten the gusset plate to it using a bolt:

Notching brace flange (60)

Some components give you the option to notch the brace flange. You may want to do this:

- if the gusset plate collides with the brace flange
- to create slots in hollow braces

To notch the flange or create a slot, go to the **Brace conn** tab. The default option is not to notch:

Creating slots in hollow braces (60)

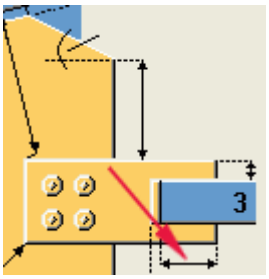
To create a slot in a hollow brace, go to the **Brace conn** tab:

1. Define the position of the gusset plate as 'Middle'. This is the default option. See Defining gusset plate position on the brace (page 3182).
2. Select this option to create a slot in the brace and fasten the gusset plate to it using a bolt:

Notching the brace (22, 59)

To define the depth of the notch in the brace, go to the **Picture** tab:

Depth of the notch in the brace



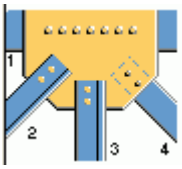
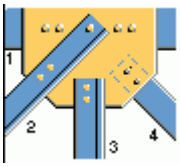
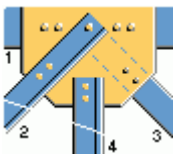
TIP To prevent the connection plate from penetrating the hollow brace, enter a negative value for the depth of the notch.

Cutting braces (60)

To indicate whether to cut the braces, go to the **Brace conn** tab. The default is to cut the braces:

Double-bolting braces (110)

By default braces are bolted to the gusset plate. To also bolt braces to the first or second beam or column picked, go to the **Brace conn** tab. The options are:

Option	Description
	Only bolts braces to the gusset plate.
	Bolts the first brace picked to both the gusset plate and the first beam or column picked.
	Bolts the first and subsequent braces picked to both the gusset plate and the first beam or column picked.

Using clip angles to connect braces (11, 57)

To attach the braces to the gusset plate using clip angles:

1. First, create a slot in the brace. To do this, go to the **Gusset** tab. The default option is:

To create a slot in the brace:

- Define the position of the gusset plate as 'Middle'. See Defining chamfer dimensions.
- Select this option to create the slot and fasten the gusset plate to the brace using a bolt:

- Go to the **Angle bolts** tab. The default option is not to create clip angles:

Creating clip angles or shear tabs (58, 61, 62, 63)

To attach the braces to the gusset plate using clip angles or shear tabs, and specify the number of clip angles to create, go to the **Brace conn** tab. The default option is to create two clip angles below the brace web:

Defining minor plate properties

As well as gusset plates, many bracing components create *minor plates*, including connection, filler, and shim plates. This section explains how to define the properties of some of these minor plates.

Click the links below to find out more:

- Defining the number of connection plates (58, 61, 62, 63) (page 3189)
- Defining connection plate width (59) (page 3189)
- Creating filler plates (58, 61, 62, 63) (page 3189)
- Creating filler plates (165) (page 3189)
- Creating shim plates (58, 61, 62, 63) (page 3190)
- Defining plate sizes in marketsizes.dat (165, Japan) (page 3190)

- Defining rib plate dimensions (165) (page 3191)
- Omitting stiffener plates (1065) (page 3191)
- Creating rib plates (22, Japan) (page 3192)

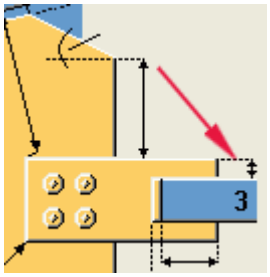
Defining the number of connection plates (58, 61, 62, 63)

To define the position and number of connection plates to create between the brace web and the gusset plate, go to the **Brace conn** tab. The default option is one plate:

Defining connection plate width (59)

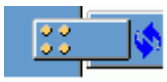
To define the width of the connection plate go to the **Picture** tab and indicate the distance from the edge of the brace to the edge of the connection plate:

This dimension defines the width of the connection plate.



Creating filler plates (58, 61, 62, 63)

To create a filler plate between the connection plate and the brace web, go to the **Brace conn** tab. The default option is no filler plate:



To create a filler plate, select this option:

Creating filler plates (165)

If required, **Heavy brace (165)** automatically creates filler plates between the connection plate and the:

- brace flange

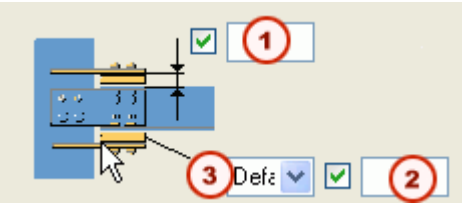
- brace web

The thickness of the filler plates depends on the gap between the connection plate and the brace flange or web, as follows:

Gap (mm)	Flange filler plate thickness	Web filler plate thickness
1	No plate	No plate
2	2.3	1.6
3	3.2	1.6
4	4.5	2.3
5	4.5	2.3
6	6.0	3.2
7	6.0	3.2
8	9.0	4.5
9	9.0	4.5
10	9	4.5

Creating shim plates (58, 61, 62, 63)

To create and define shim plates, go to the **Brace conn** tab. The options are:



	Description
1	Clearance. Enter 1 mm or more to create shim plates.
2	Enter number of shim plates to create.
3	Select **No** to omit shim plates.

Defining plate sizes in marketsizes.dat (165, Japan)

In the Japanese environment, **Heavy brace gusset (165)** takes the default thickness values for several plates from the `marketsizes.dat` file, located in the `../profil/` folder. This file lists the standard plate thicknesses available

in specific material grades. Edit this file using any standard text editor, such as Notepad.

NOTE Enter a value in the **t** field on the **Parts** tab to override the default value for any plate.

Example

In this example, the `marketsizes.dat` file lists the standard plate thicknesses available in material grade SS400. The DEFAULT line lists the thicknesses available in all other material grades.

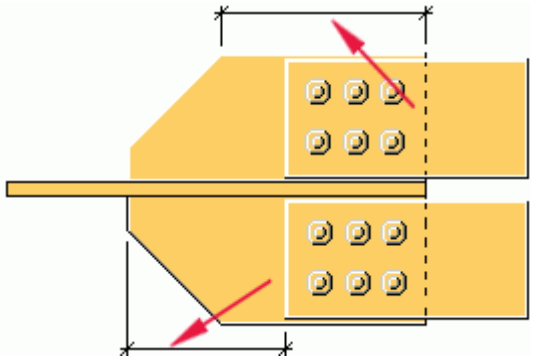
```
# Market size (thickness) table
# Specify the material as given in Tekla
    Structures material database
# and after that the market sizes separated by
    commas
SS400,1.6,2.3,3.2,4.5,6,9,12,16,19,22,25,28,32
    ,38
DEFAULT,6,9,12,16,19,22,25,28,32,38
```

marketsizes.dat file

Defining rib plate dimensions (165)

Tekla Structures automatically calculates the following dimensions from the length of the rib plate and bolt group properties.

These dimensions are automatically calculated from the length of the rib plate and the bolt group properties:



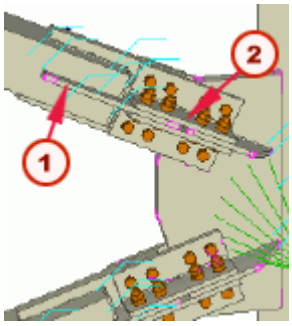
To define the remaining dimensions, go to the **Ribs** tab.

Omitting stiffener plates (1065)

This component automatically creates stiffener plates at each end of the gusset plate. To omit one or both stiffener plates, go to the **Parts** tab and delete the values in the **Front stiffener** or **Back stiffener** fields.

Creating rib plates (22, Japan)

To create this type of connection:



	Description
1	Cross plate
2	Cover plate

Go to the **Rib plates** tab and enter properties in the **Cross plate** and **Cover plate** fields.

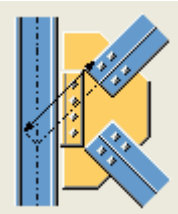
Defining bolt and hole properties

Click the links below to find out more:

- Brace bolt properties (11, 57) (page 3192)
- Defining part thickness (1) (page 3193)
- Defining hole type (1) (page 3193)
- Dimensioning holes in drawings (110, 140) (page 3194)
- Position of gusset bolts (11) (page 3194)

Brace bolt properties (11, 57)

To define the minimum distance from the connection plate bolts to the intersection point of the column and brace centerlines, go to the **Bracebolts** tabs:

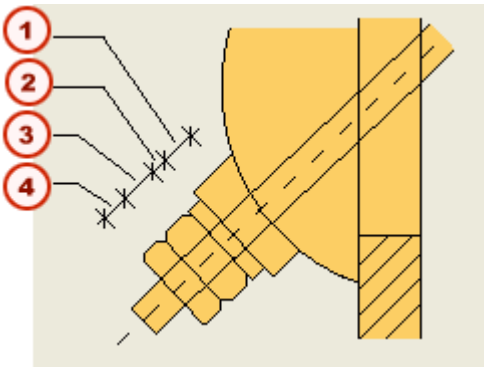


The gap is measured parallel to each brace. If a brace is perpendicular to the column, the gap is measured from the column centerline to the nearest bolts:

Defining part thickness (1)

To define the thickness of the filler plate, washer, and nuts that this component creates, go to the **Parts** tab and enter the following dimensions:

Enter the thickness of the following parts:



	Description
1	Filler plate
2	Washer
3	Nut
4	Extra nut

Defining hole type (1)

To define the type of hole to create in the beam or column for the rod, go to the **Brace conn** tab and select one of the options in the **Create hole in primary** field:

- **Partcut** cuts a hole for the rod

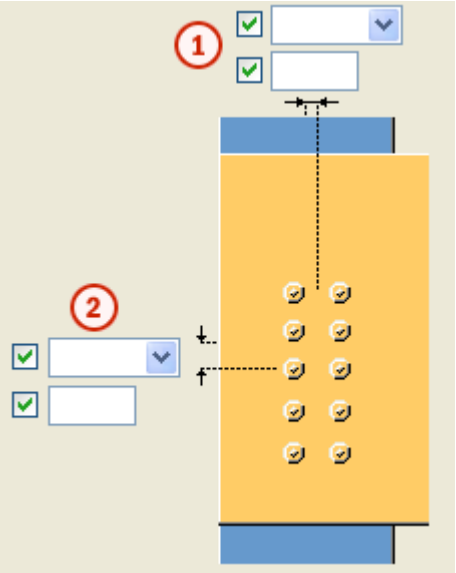
- **Bolt** creates a bolt hole
- **Partcut+bolthole** cuts a hole for the rod and creates a bolt hole

Dimensioning holes in drawings (110, 140)

If a component creates a gusset plate that is skewed so that the gusset hole is not perpendicular to the gusset plate, hole dimensions in drawings may be wrong. To fix this, go to the **Gusset** tab and set the **Create extra bolt holes to the gusset** field to **Yes**. This creates an extra bolt hole that is perpendicular to the gusset plate.

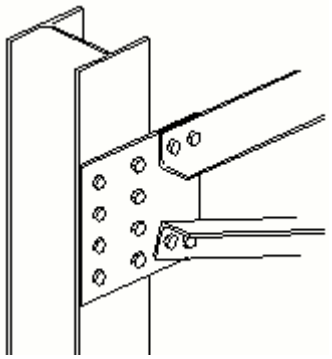
Position of gusset bolts (11)

To find the right position for the gusset bolt group, go to the **Gusset bolts** tab. The position options are for the horizontal (1) and vertical (2) direction:



Horizontal position

Gusset horizontal position and offset options (1) only affect the following situation, where the gusset is connected to the column flange. When the position is **Middle**, the bolt group offset is calculated from the center line of the column, for example.



Vertical position

Gusset vertical position and offset options (2) affect the bolt group connecting the gusset directly to the flange and also the bolt groups connecting the gusset to the clip angle or connection plate.

Defining other properties

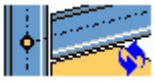
Click the links below to find out more:

- Specifying work point location (58, 59, 60) (page 3195)
- Defining Tees (105) (page 3195)

Specifying work point location (58, 59, 60)

Tekla Structures uses the *work point* of a component to calculate check dimensions and part position dimensions in drawings.

To change the position of the work point, go to the **Picture** tab. The default position is the point where the two main parts intersect:



Defining Tees (105)

There are several ways to create the tees at the end of the braces:

- On the **Parts** tab, specify a T profile to use.
- On the **Parts** tab, specify an I profile to use. The component creates the tee by cutting the I profile. You must specify where to cut the I profile, using the **Tee depth** field on the **Parameters** tab.
- To have Tekla Structures use two plates to create the tee, leave the **Tee profile** field on the **Parts** tab blank. You must enter **t,b**, and **h** dimensions for the tee flange and web.

NOTE You can define the Tees for each brace separately.

5.18 Tower components

You can use tower components to automatically create:

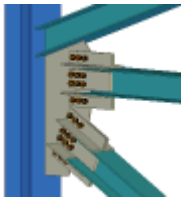
- Complete towers, and tower elements such as legs and bracing
- Brace to tower leg connections
- Brace to brace connections

See also

Tower elements (page 3196)
Brace to tower leg connections (page 3202)
Brace to brace connections (page 3209)
Editing tools (page 3216)

Tower elements

Tekla Structures includes the following components that you can use to automatically create a complete tower, or structural elements, such as tower legs or bracing:

Component	Icon	Description
Tower generation (S43) (page 3197)		Creates a complete tower.
Tower member (S63) (page 3198)		Creates tower legs.
Transmission tower cross arm (S65) (page 3200)		Creates cross arms using bent, angle-profile cleats.

Component	Icon	Description
Tower diagonal (S66) (page 3201)		Creates bracing panels.

Tower generation (S43)

Creates a complete tower, with a square or rectangular base.

Profiles

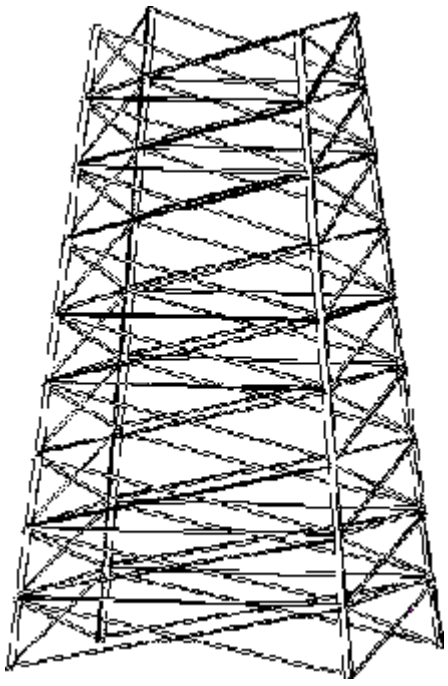
Tower legs: L

Bracing: L, flat, U, twin

Parts created

- Tower legs (4)
- Bracing panels (you define the quantity)

Use for

Situation	Description
	

Before you start

Check the current work plane, as it affects the position of the tower. For more information, read Position of the tower (S43, S63) (page 3221)

Defining properties

Use the following tabs in the component dialog box to define the properties of the parts this component creates:

Tab	Contents	See also
Picture	Quantity of bracing panels, dimensions that define the location of cross braces on the tower legs.	Defining bracing panels (S43, S66) (page 3227)
Parts	Part properties, profiles for legs and bracing.	
Leg Parameters	Quantity of profiles in each leg, lift length, option to tile profiles	Defining tower legs (S63) (page 3224)
Tower Parameters	Options that define the type of bracing to create, class numbers of bracing.	Defining bracing panels (S43, S66) (page 3227)
Twin Profiles	Options to use twin profiles for bracing.	
Model Points		Creating construction points (S43, S66) (page 3221)
Joints	Components that connect braces to tower legs.	Defining bracing connections (S43, S66) (page 3227)

Picking order

Pick a point to indicate the position of the tower leg at the bottom left corner of the base of the tower.

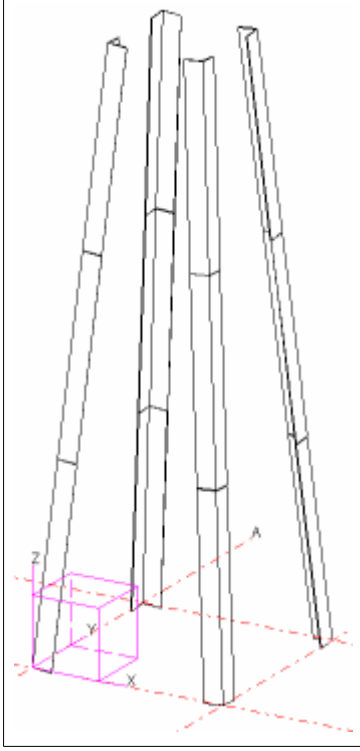
Tower member (S63)

Creates the 4 legs of a tower, using angle profiles. The legs form a square or rectangular base.

Parts created

Legs (4)

Use for

Situation	Description
	

Before you start

Check the current work plane, as it affects the position of the tower. For more information, read Position of the tower (S43, S63) (page 3221)

Defining properties

Use the following tabs in the component dialog box to define the properties of the parts this component creates:

Tab	Contents	See also
Picture	Properties of angle profiles, quantity of profiles in each leg, lift length, option to tile profiles.	Defining tower legs (S63) (page 3224) Creating sloping legs (S63) (page 3225)
Parts		
Parameters	Gaps between angle profiles	Pattern of angle profiles (S63) (page 3225)

Picking order

Pick a point to indicate the lower left corner of the tower.

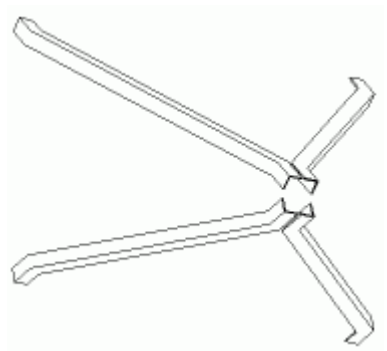
Transmission tower cross arm (S65)

Creates cross arms using bent, angle-profile cleats.

Parts created

Bent, angle-profile cleats

Use for

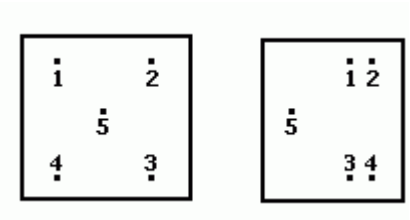
Situation


Do not use for

Twin-profile cross arms.

Before you start

Create 5 points. The 5th point must be in the center of the pattern, vertically:



Defining properties

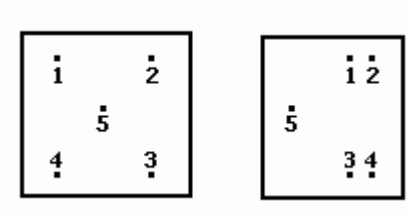
Use the following tabs in the component dialog box to define the properties of the parts this component creates:

Tab	Contents	See also
Picture	Dimensions that define the shape and size of the upper and lower cross arms.	
Parts	Part properties of the cross arms.	

Tab	Contents	See also
Parameters	Layout of profiles	Layout of profiles (S65) (page 3226)

Picking order

The picking order depends on the location of the 5th point:



Tower diagonal (S66)

Creates bracing panels between 2 or 4 existing columns.

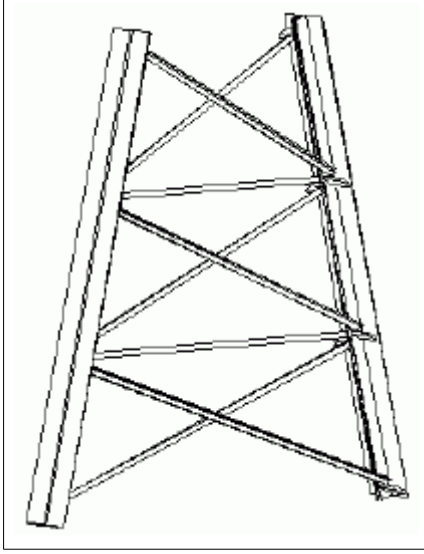
Profiles

Bracing: L, flat, U, twin

Parts created

- Horizontal braces
- Diagonal braces

Use for

Situation	Description
	

Before you start

Create 2 or 4 tower legs.

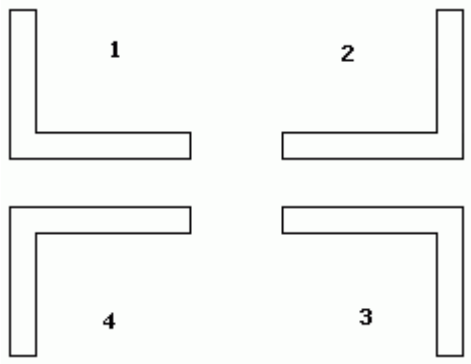
Defining properties

Use the following tabs in the component dialog box to define the properties of the parts this component creates:

Tab	Contents	See also
Picture	Dimensions that define the location of diagonal braces, quantity of bracing panels.	Defining bracing panels (S43, S66) (page 3227)
Parts	Part properties of the horizontal and diagonal braces.	
Parameters	Options that define the type of bracing to create.	Defining bracing panels (S43, S66) (page 3227)
Twin Profiles	Option to create twin-profile braces, position of twin profiles.	
Model Points	Option to create construction points, construction point properties.	Creating construction points (S43, S66) (page 3221)
Joints	Components to use to connect braces to tower legs.	Defining bracing connections (S43, S66) (page 3227)

Picking order

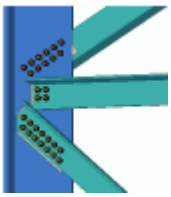
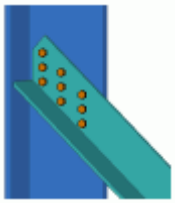
1. Pick the tower legs clockwise, starting with the top left leg:



2. Click the middle mouse button to create the component

Brace to tower leg connections

Tekla Structures includes the following components to connect one or more braces to tower legs:

Component	Icon	Description
Tower 1 diagonal (87) (page 3203)		Bolts 1 diagonal brace to a tower leg.
Tower 2 diagonal (89) (page 3205)		Bolts 2 diagonal braces to a tower leg.
Leg - 2 and 3 diagonals (177) (page 3206)		Bolts 2 diagonal braces and 1 horizontal brace (optional) to a tower leg. No gusset plate.
Leg - 1 diagonal (178) (page 3208)		Bolts 1 diagonal brace directly to the outside or inside face of a tower leg. No gusset plate.

Tower 1 diagonal (87)

Bolts 1 diagonal brace to a tower leg.

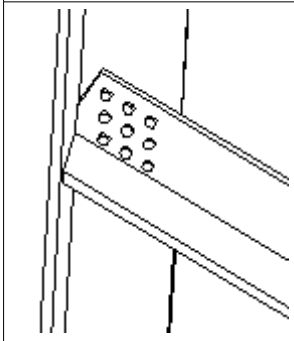
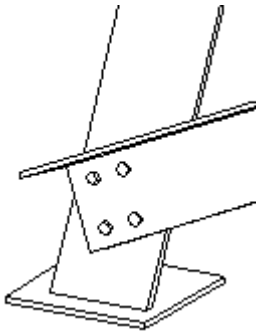
Profiles

Brace and tower leg: L

Parts created

-

Use for

Situation



Before you start

Create a tower leg and a brace.

Defining properties

Use the following tabs in the component dialog box to define the properties of the parts this component creates:

Tab	Contents	See also
Picture	Option to create bolts, Location of bolts.	About bolt gage lines (page 3233) Creating bolts (89) (page 3236)
Parts	Option to cut vertical leg of brace, dimensions of cut.	Cutting braces (87, 89) (page 3228)
Bolts	Bolt properties	
General	Connection properties, AutoDefaults and AutoConnection rule groups.	

Tab	Contents	See also
Extra cuts	Option to cut horizontal leg of brace, dimensions of cut.	
Check		
Analysis	Information used in structural analysis	

Selection order

1. Tower leg
2. Brace

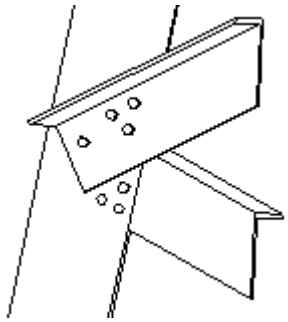
Tower 2 diagonal (89)

Bolts 2 diagonal braces to a tower leg.

Parts created

-

Use for

Situation	More information
	

Do not use for

Braces that clash.

Before you start

Create a tower leg and 2 diagonal braces.

Defining properties

Use the following tabs in the component dialog box to define the properties of this component:

Tab	Contents	See also
Picture	Bolt gages that define the location of bolts that connect all parts.	About bolt gage lines (page 3233) Creating bolts (89) (page 3236)
Picture2	Bolt gages that define the location of bolts in the bolt groups that connect: • 1st brace to the tower leg • 2nd brace to the tower leg	
Parts	Dimensions that define the cuts in the vertical legs of the diagonal braces.	Cutting braces (87, 89) (page 3228)
General	Connection properties, AutoDefaults and AutoConnection rule groups.	
Cuts d.1	Dimensions that define the cuts in the horizontal leg of the first brace picked.	Cutting braces (87, 89) (page 3228)
Cuts d.2	Dimensions that define the cuts in the horizontal leg of the second brace picked.	
Bolts	Bolt properties.	
Check		
Analysis	Information used in structural analysis	

Selection order

1. Tower leg
2. First brace
3. Second brace
4. Click the middle mouse button to create the component

Leg - 2 and 3 diagonals (177)

Bolts 2 diagonal braces and 1 horizontal brace (optional) to a tower leg. Does not create a gusset plate.

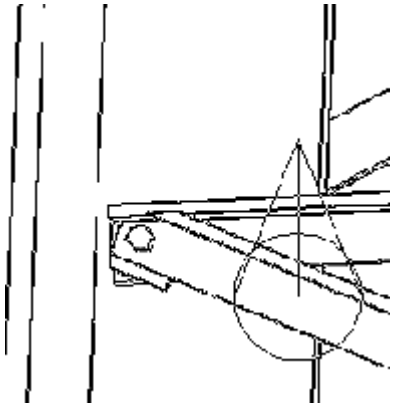
Profiles

Tower leg and braces: L

Parts created

Filler plates (optional)

Use for

Situation


Before you start

Create the following parts

- Tower leg
- 1 diagonal brace to the outside of the tower leg
- 1 diagonal brace to the inside of the tower leg
- 1 horizontal brace (optional)

Defining properties

Use the following tabs in the component dialog box to define the properties of the parts this component creates:

Tab	Contents	See also
Picture	Edge distances, dimensions that define the cut in the diagonal brace on the inside of the tower leg and the horizontal brace.	Creating your own defaults (177) (page 3230)

Tab	Contents	See also
Parameters	Edge distances that define the location of bolt groups.	Creating your own defaults (177) (page 3230)
Bolts	Bolt properties, dimensions that define the location of bolts, and bolt pattern of bolt groups.	
General	Connection properties, AutoDefaults and AutoConnection rule groups.	
Plates	Option to create filler plates, filler plate properties.	Defining filler plates (177) (page 3239)
Design	Option to use UDL with AutoDefaults, reaction forces.	
Analysis	Information used in structural analysis	

Selection order

1. Tower leg
2. Diagonal brace on the inside of the tower leg
3. Diagonal brace on the outside of the tower leg
4. Horizontal brace (optional)
5. Click the middle mouse button to create the component

Leg - 1 diagonal (178)

Bolts 1 diagonal brace directly to the outside or inside face of a tower leg.

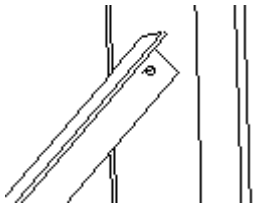
Profiles

Brace and tower leg: L

Parts created

-

Use for

Situation	More information
	

Before you start

Create a tower leg and a diagonal brace.

Defining properties

Use the following tabs in the component dialog box to define the properties of the parts this component creates:

Tab	Contents	See also
Picture	Dimensions that define how the brace is cut.	
Parameters	Bolt gage lines that define the positions of bolts.	About bolt gage lines (page 3233)
Bolts	Quantity of bolts, bolt properties.	
General	Connection properties, AutoDefaults and AutoConnection rule groups.	
Design	Option to use UDL with AutoDefaults, reaction forces.	
Analysis	Information used in structural analysis	

Selection order

1. Tower leg
2. Diagonal brace

Brace to brace connections

Component	Icon	Description
Bolted gusset brace (167) (page 3210)		Bolts 2 diagonal braces to 2 horizontal braces, using a gusset plate.
Bolted bridge brace (169) (page 3211)		Bolts 2 horizontal braces and 1 diagonal brace to a bridge plate or angle profile.
Bolted Brace (181) (page 3213)		Bolts 1 diagonal brace to 1 or 2 horizontal braces.
Bolted Plate Brace (182) (page 3215)		Bolts 1 diagonal brace to 1 or 2 horizontal braces, using a plate.

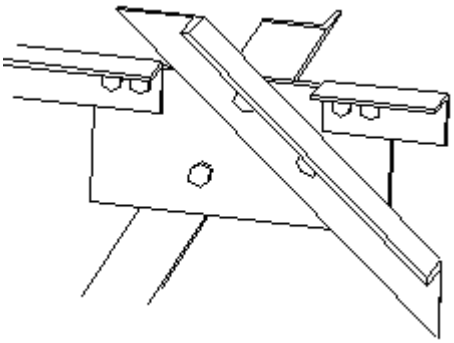
Bolted gusset brace (167)

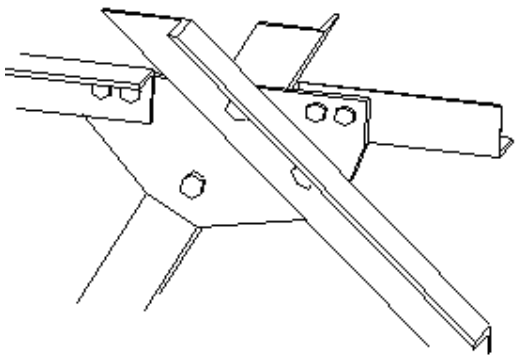
Bolts 2 diagonal braces to 2 horizontal braces, using a gusset plate.

Parts created

Gusset plate

Use for

Situation	Description
	Rectangular gusset plate.

Situation	Description
	Braces connect to different faces of chamfered gusset plate.

Before you start

Create 2 diagonal braces and 2 horizontal braces.

Defining properties

Use the following tabs in the component dialog box to define the properties of the parts this component creates:

Tab	Contents	See also
Picture	Dimensions that define bolt locations.	About bolt gage lines (page 3233)
Parts	Properties of gusset plate.	
Parameters	Properties of bolts in horizontal braces, option to chamfer gusset plate.	
General	Connection properties, AutoDefaults and AutoConnection rule groups.	
Analysis	Information used in structural analysis	

Selection order

1. First diagonal brace
2. Second diagonal brace
3. First horizontal brace
4. Second horizontal brace
5. Click the middle mouse button to create the component

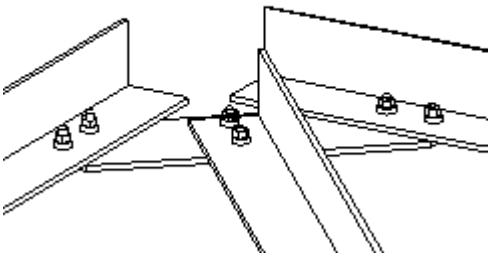
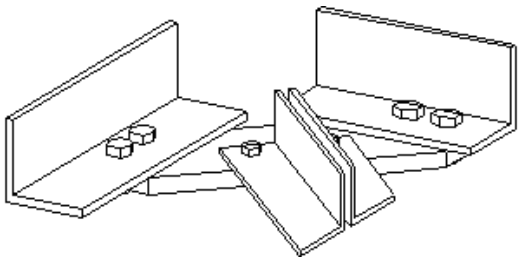
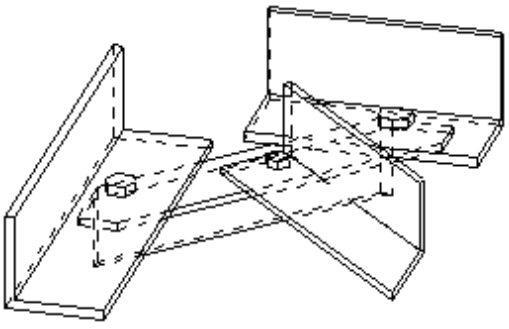
Bolted bridge brace (169)

Bolts 2 horizontal braces and 1 diagonal brace to a bridge plate or angle profile. Cuts the diagonal brace.

Parts created

Bridge plate or angle profile

Use for

Situation	More information
	
	Twin-profile diagonal braces
	Angle profile used as a bridge

Do not use for

Braces on different planes, or braces that clash.

NOTE **Bolted bridge brace (69)** does not cut the horizontal braces.

Before you start

Create 2 horizontal braces and 1 diagonal brace (single or twin profile).

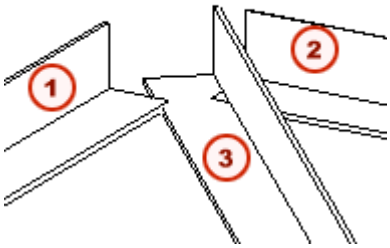
Defining properties

Use the following tabs in the component dialog box to define the properties of the component:

Tab	Contents
Picture	Dimensions that define bolt locations and the clearance between the bridge and horizontal braces.
Parts	Part properties of the bridge.
Parameters	Properties of bolts in horizontal braces, bridge cut options.

Picking order

1. First horizontal brace
2. Second horizontal brace
3. Diagonal brace
4. If the diagonal brace is a twin profile, pick the second profile.
5. Click the middle mouse button to create the component.



	Description
1	First horizontal brace
2	Second horizontal brace
3	Diagonal brace

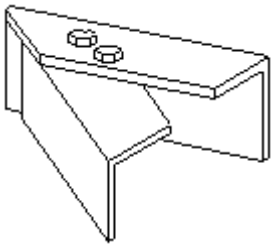
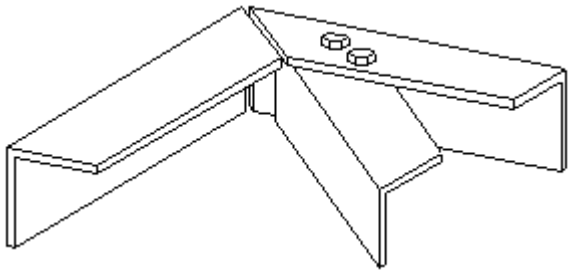
Bolted Brace (181)

Bolts 1 diagonal brace to 1 or 2 horizontal braces.

Parts created

-

Use for

Situation



Do not use for

Braces that are not in the same plane.

Before you start

Create 1 diagonal brace and 1 or 2 horizontal braces. The braces can be built-up beams.

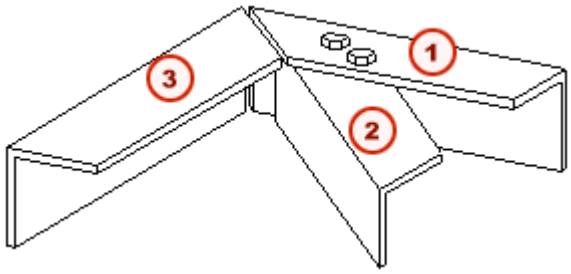
Defining properties

Use the following tabs in the component dialog box to define the properties of this component:

Tab	Contents	See also
Picture	Dimensions that define the position of bolts. Clearance between braces.	Defining bolt gage lines (87) (page 3234) Cutting braces (181, 182) (page 3229)
Parameters	Bolt and cut options for horizontal braces.	

Picking order

1. First horizontal brace
2. Diagonal brace
3. Second horizontal brace (optional)



	Description
1	First horizontal brace
2	Diagonal brace
3	Second horizontal brace (optional)

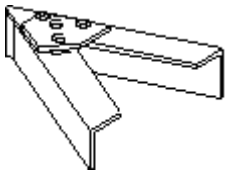
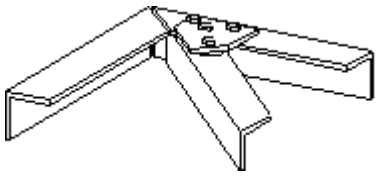
Bolted Plate Brace (182)

Bolts 1 diagonal brace to 1 or 2 horizontal braces, using a plate. Connects the diagonal brace to the inside or outside face of the horizontal brace.

Parts created

- Plate
- Filler plate(s)

Use for

Situation



Do not use for

Braces that are not in the same plane.

Before you start

Create 1 diagonal brace, and 1 or 2 horizontal braces.

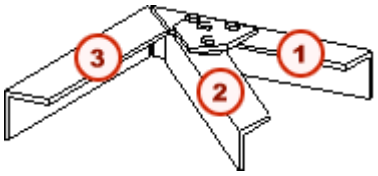
Defining properties

Use the following tabs in the component dialog box to define the properties of the parts this component creates:

Tab	Contents	See also
Picture	Dimensions that define the location of bolts, clearance between horizontal braces.	About bolt gage lines (page 3233) Cutting braces (181, 182) (page 3229)
Parts	The properties of the plate and optional filler plate.	
Parameters	Shape of cut in braces, bolt options, plate options.	Defining filler plates (182) (page 3240) Creating bolts (182) (page 3237)

Picking order

1. Horizontal brace to which to bolt the diagonal brace
2. Diagonal brace
3. Second horizontal brace (optional)
4. Click the middle mouse button to create the component



	Description
1	Horizontal brace to which to bolt the diagonal brace
2	Diagonal brace
3	Second horizontal brace (optional)

Editing tools

Use the following components to modify tower bracing:

Component	Icon	Description
Open/Close angle ends		Simulates the opening or closing of one end of an angle profile.

Component	Icon	Description
(1050) (page 3217)		
Open/Close angle (1051) (page 3218)		Simulates the opening or closing of an inner portion of an angle profile.
Autoposition (S67) (page 3220)		Adjusts the position of braces connected to a tower leg. Cuts braces.

Open/Close angle ends (1050)

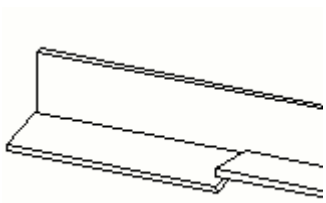
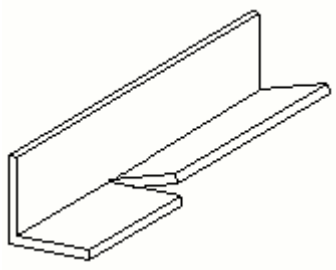
Simulates the opening or closing of one end of an angle profile by cutting and bending the profile.

NOTE Use this component to create detail drawings that indicate where to open or close angle profiles used in tower bracing. In the shop, the angle profile is machined, not cut.

Parts created

-

Use for

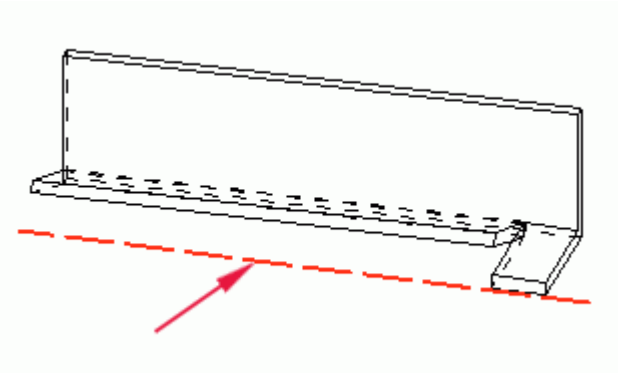
Situation	More information
	
	

Do not use for

To open or close the inner portion of an angle profile, use Open/Close angle (1051) (page 3218).

Before you start

- Create an angle profile
- Create a point to locate the component
- Set the work plane parallel to the outer face of the leg of the angle profile that you want to bend:



Defining properties

Use the following tab in the **Open/Close angle ends (1050)** dialog box to define the properties of the component:

Tab	Contents	More information
Parameters	Which leg to open or close, location of cut, bend angle.	Adjusting length of leg to open or close (1050, 1051) (page 3222)

Picking order

1. Angle profile
2. A point on the angle profile

Open/Close angle (1051)

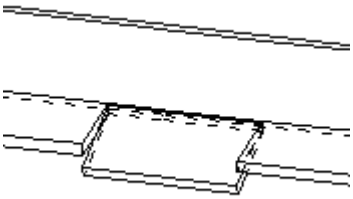
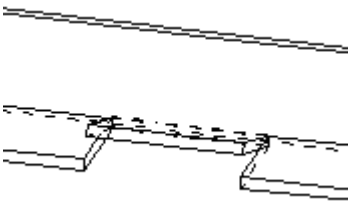
Simulates the opening or closing of an inner portion of an angle profile by cutting and bending the profile.

NOTE Use this component to create detail drawings that indicate where to open or close angle profiles used in tower bracing. In the shop, the angle profile is machined, not cut.

Parts created

-

Use for

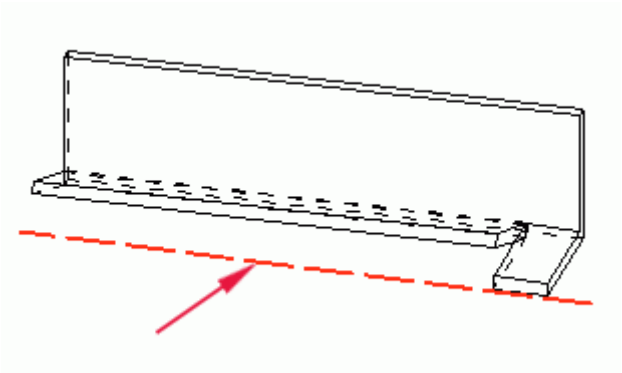
Situation	More information
	
	

Do not use for

To open or close one end of an angle profile, use Open/Close angle ends (1050) (page 3217).

Before you start

- Create a point to locate the component
- Set the work plane parallel to the outer face of the leg of the angle profile that you want to bend:



Defining properties

Use the following tabs in the component dialog box to define the properties of the component:

Tab	Contents	More information
Parameters	Which leg to open or close, location of cut, bend angle.	Adjusting length of leg to open or close (1050, 1051) (page 3222)

Picking order

1. Angle profile
2. A point on the angle profile

Autoposition (S67)

Adjusts the position of 1 or 2 braces connected to a tower leg. Cuts braces.

WARNING Use **Autoposition (S67)** when you have completed all other work on a model. Changing the model may override the adjustments you make using this component.

Parts created

-

Before you start

Create 1 or 2 braces and connect them to a tower leg.

Defining properties

Use the following tab in the component dialog box to adjust the braces and create cuts:

Tab	Contents	See also
Picture		Moving and cutting braces (S67) (page 3231)

Picking order

1. Tower leg
2. First brace
3. Second brace (optional)
4. Click the middle mouse button

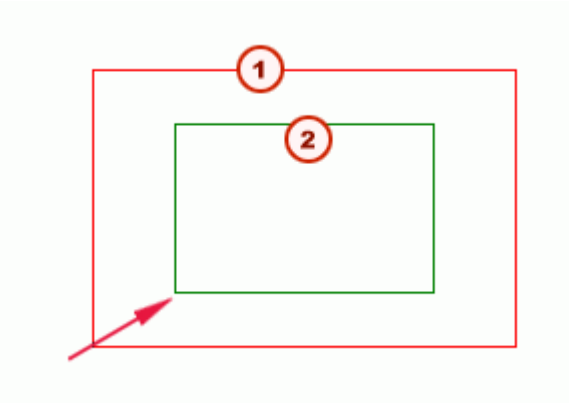
Defining general properties

Click the links below to find out more:

- Position of the tower (S43, S63) (page 3221)
- Creating construction points (S43, S66) (page 3221)
- Adjusting length of leg to open or close (1050, 1051) (page 3222)

Position of the tower (S43, S63)

Tekla Structures creates the tower along the z axis of the current work plane. If the tower has a rectangular base, the longest side of the base runs parallel to the x axis:



	Description
1	Base of tower
2	Top of tower

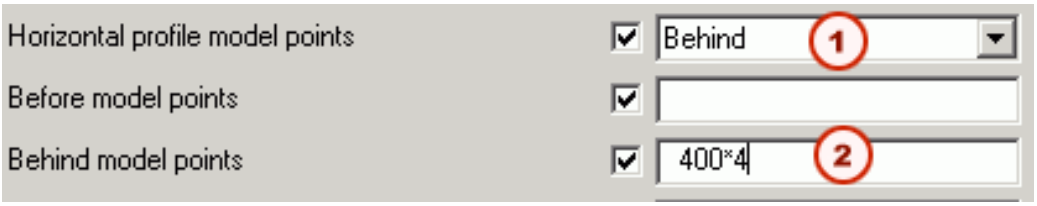
Creating construction points (S43, S66)

Construction points are points that you can pick to attach components to parts. For example, you might create construction points on tower legs to use to connect bracing to the legs.

To create construction points on tower legs or braces, go to the **Model Points** tab. For each type of brace:

- Select the location of the points (1). For example, select **Before** to create points on the outer face of a brace;

- Enter the distance between the points, followed by the quantity of points (2). For example, enter 400*4 to create 4 points, 400 mm apart.



Adjusting length of leg to open or close (1050, 1051)

To adjust the length of leg to open or close, go to the **Parameters** tab and enter one of the following dimensions:

Field	Description
External length of the portion to be opened/closed	Measured from the point you pick to create the component towards the start of the part reference line
Internal length of the portion to be opened/closed	Measured from the point you pick to create the component towards the end of the part reference line

Defining tower leg properties

This section explains how to define the properties of tower legs.

Click the links below to find out more:

- Defining tower legs (S43) (page 3222)
- Defining tower legs (S63) (page 3224)
- Layout of profiles (S65) (page 3226)

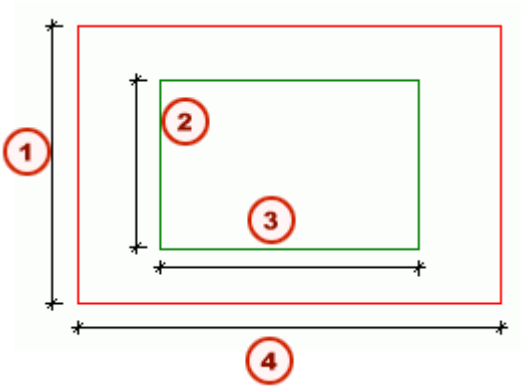
Defining tower legs (S43)

Tower legs are constructed using angle profiles.

- Creating sloping legs (S43) (page 3222)
- Type and quantity of angle profiles (S43) (page 3223)

Creating sloping legs (S43)

To define the slope of the tower legs, go to the **Leg Parameters** tab and enter the x and y dimensions of the base and top of the tower:

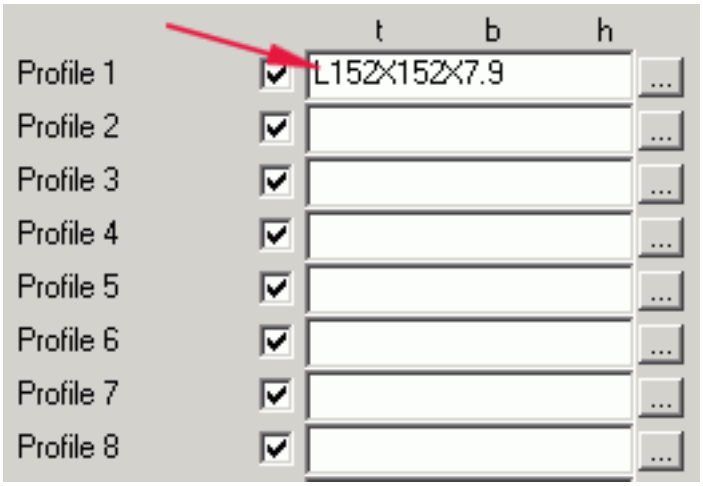


	Description
1	Base y
2	Top y
3	Top x
4	Base x

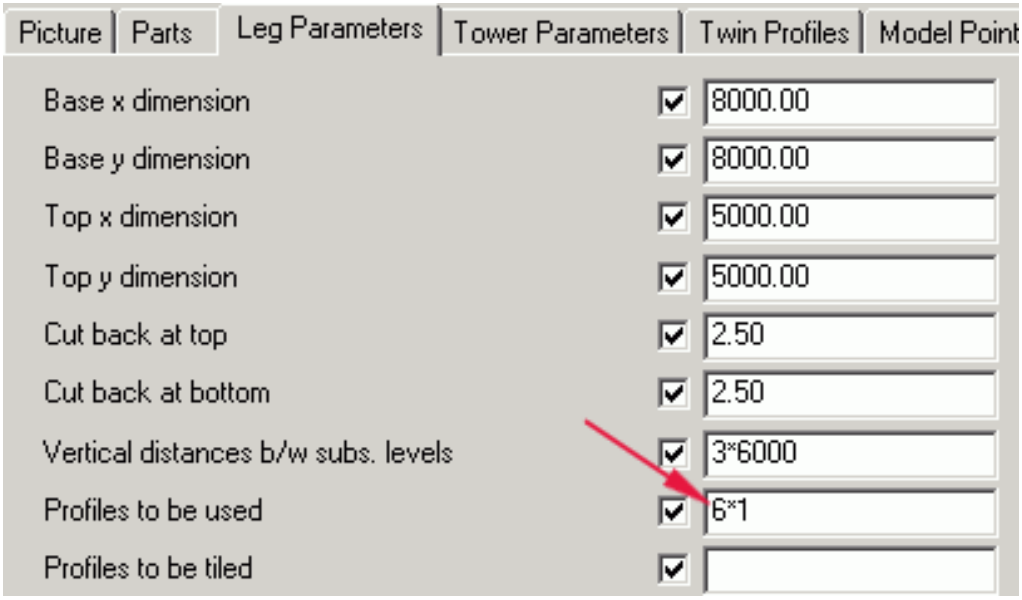
Type and quantity of angle profiles (S43)

To specify the angle profiles to use for the tower legs:

1. Go to the **Parts** tab and enter the profiles you want to use in the **Profile 1**, **Profile 2**, etc. fields. You can specify up to 8 profile types.



2. Go to the **Profiles to be used** field on the **Leg Parameters** tab and enter the quantity of each profile type to use for the legs. Here we are using 6 lifts of Profile 1 to create the tower leg:



Defining tower legs (S63)

Tower legs are constructed using angle profiles.

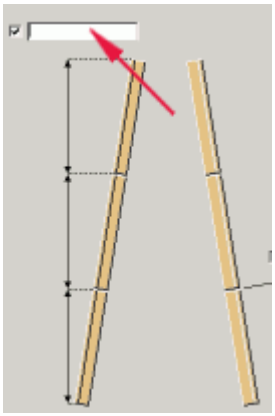
- Type and quantity of angle profiles (S63) (page 3224)
- Creating sloping legs (S63) (page 3225)
- Pattern of angle profiles (S63) (page 3225)
- Splicing legs (S63) (page 3226)

Type and quantity of angle profiles (S63)

To define the type and quantity of angle profiles to use:

1. Go to the **Picture** tab. Use fields **1** to **8** to define the angle profiles you want to use.
2. In the **Profiles for part** field, enter the quantity of each profile type to use for each leg. For example, enter 6*1 to create tower legs each made up of 6 lifts with the profile type you defined in field **1**.

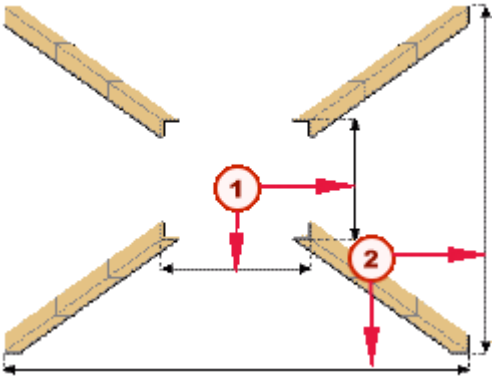
3. On the illustration, enter the quantity and length of the lifts that make up each leg. Enter 6*6000 to create tower legs using 6 lifts, each 6000 long:



4. To tile specific leg lifts, enter the numbers of the lifts to tile in the **Parts to be tiled** field, counting from the bottom of the leg. For example, enter 3 5 to tile lifts 3 and 5.

Creating sloping legs (S63)

To create sloping tower legs, go to the illustration on the **Parts** tab and enter the x and y dimensions of the base and top sections of the tower:



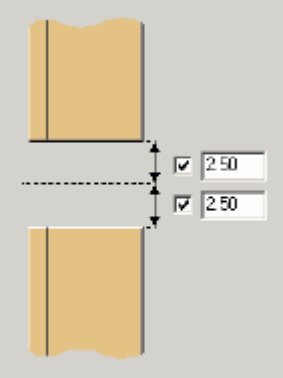
	Description
1	Dimensions of top of tower
2	Dimensions of base of tower

Pattern of angle profiles (S63)

To define the quantity of angle profiles that form each leg, in cross section, use the graphic option on the **Parts** tab. The default option is one angle profile:

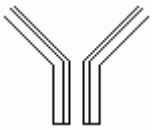
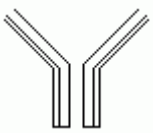
Splicing legs (S63)

To enable the erector to splice tower legs, go to the **Parameters** tab and set the cut back distance between the angle profiles:



Layout of profiles (S65)

The options are:

Option	Layout
Inside	
Outside	

Defining tower bracing properties

This section explains how to define the properties of tower bracing.

Click the links below to find out more:

- Defining bracing panels (S43, S66) (page 3227)
- Defining bracing connections (S43, S66) (page 3227)
- Cutting braces (87, 89) (page 3228)
- Cutting braces (177) (page 3229)
- Cutting braces (181, 182) (page 3229)
- Creating your own defaults (177) (page 3230)
- Moving and cutting braces (S67) (page 3231)

Defining bracing panels (S43, S66)

To define the quantity of bracing panels to create between each pair of tower legs, enter a number in the **Number of Diagonals field** on the **Picture** tab.

To define the layout of the bracing panels, go to the **Tower Parameters** tab and select an option in the **Type of Diagonal** dropdown list. The default layout is cross bracing:

You can also create diagonal bracing, in various layouts.

Defining bracing connections (S43, S66)

To specify the components to use to connect braces to the tower legs, go to the **Joints** tab. You can use different components to connect the left and right diagonal braces, and the horizontal braces.

WARNING You cannot use custom components to connect the braces to the tower legs.

To define each connection, on the **Joints** tab:

- Select a component from the dropdown list,

or

- To select a component that is not on the dropdown list,, select **Custom** and enter the component number.

You can also use a predefined set of properties for the component:

1. Enter the number of the component to use (1). Here we are using **Fitting (13)**.

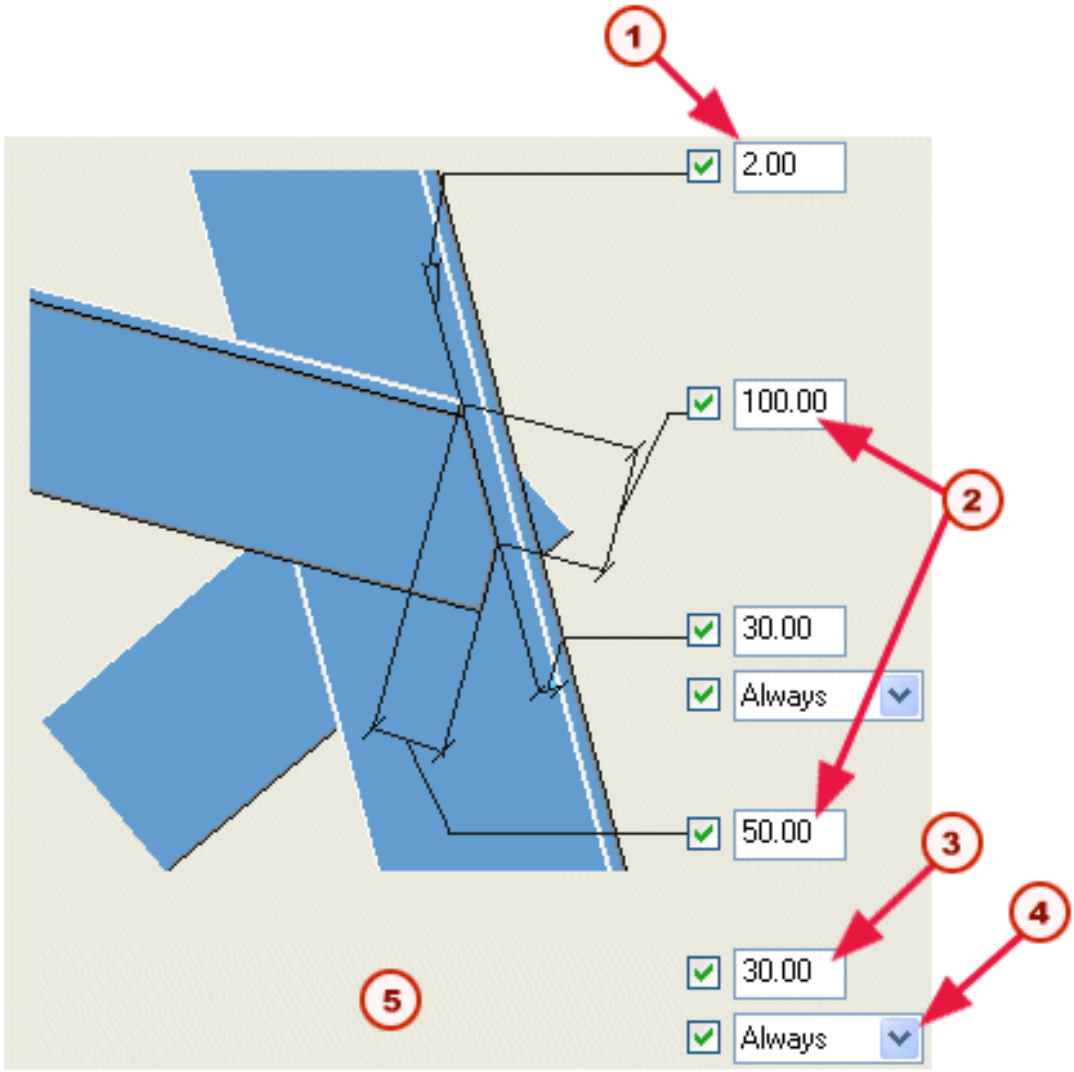
2. Indicate the predefined set of properties to use (2).

Cutting braces (87, 89)

To define the cuts in the legs of the braces, go to:

- Horizontal legs: **Parts** tab
- Vertical legs: **Extra cuts** tab (87), **Cuts d.1** and **Cuts d.2** tab (89)

These tab pages contain options to cut the legs of the braces, and the dimensions of the cut. This is how they appear on the **Parts** tab:



	Description
1	Incremental value for the angle of the cut. If you enter 2, the actual angle can be 2, 4, 8, etc
2	Cut dimensions
3	Clearance to heel of tower leg
4	Cut options
5	Clearance between main profile leg and diagonal 2

NOTE Use the **Always** cut option to cut the braces and create the clearance to the heel of the tower leg. This option overrides the bolt edge distances on the **Picture** tab.

Cutting braces (177)

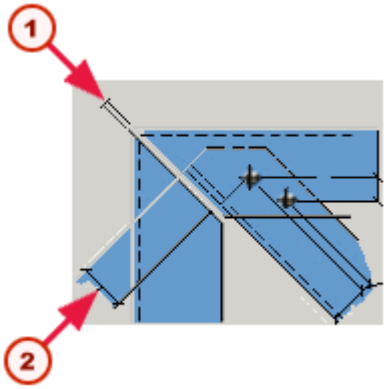
Leg - 2 & 3 diagonals (177) automatically cuts the braces according to the bolt edge distances specified for each brace on the **Picture** tab.

Cutting braces (181, 182)

Tekla Structures automatically:

- Miters the end of the horizontal brace, or braces and
- Cuts the end of the diagonal brace, according to the bolt edge distance.

To specify these dimensions, go to the **Picture** tab:



	Description
1	Clearance between horizontal braces
2	Bolt edge distance

To define the shape of the cut, use the following fields:

- **Switch for shape of horiz. braces cut** on the **Parameters** tab (181).

- **Switch to manage brace cut shape** on the **Parameters** tab (182).

Creating your own defaults (177)

Default values for all the properties in the **Parameters** tab, except clearance, plus the bolt end distances of the diagonals can be specified in text files called `tower_joint_clearance_N.txt`, where N is the bolt diameter. For example, `tower_joint_clearance_16.txt`, `tower_joint_clearance_24.txt`, etc.

This file can be created in the current model folder or system folder.

If the text file for a particular bolt diameter is not found, the connection will calculate its own default values.

The format of this text file is:

Format 1: <L-profile> <C> <D1> <D2> <D3> <D4> <E> | <C> <D1> <D2> <D3> <D4> <E>

Format 2: <L-profile> <C> <D1> <D2> <D3> <D4> <E>

Format 3: <L-profile> <H> | <B>

Format 4: <L-profile> <H>

Where

- <L-profile> = L-profile name; eg, L200*200*20, L200/15.
- <C> = distance from the edge of the part to the nearest bolt.
- <D1> = distance from the heel of the part.
- <D2> = distance from the outside-border of the part.
- <D3> = distance from the heel of the part, away from the part.
- <D4> = distance from the cut-end of the part, if any.
- <E> = distance between bolts.
- <H> = see the section below on Format 3 & 4.
- <B> = see the section below on Format 3 & 4.
- The first set of values in Format 1 is for the first flange (h) of the part, and the second set for the second flange (b).
- In Format 2, h and b have the same values.
- In Format 3, the first set of <C> <D1> ... <D4> <E> values are assigned with the value of <H>, and the second assigned with <B>.
- In Format 4, both sets of values are assigned with the value of <H>.

Note that

- Lines starting with a ';' or ' ' are skipped.
- <L-profile> must start on the first position of the line.

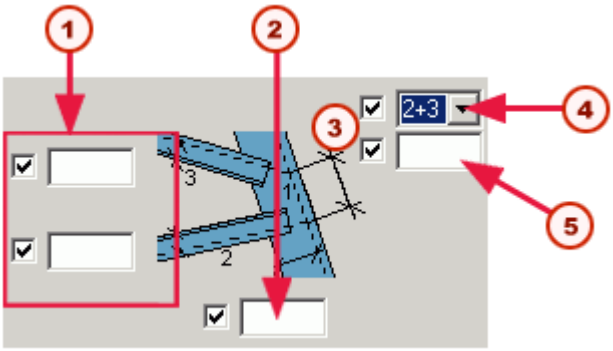
- There must be at least one blank space between <L-profile> and the first value.
- The component finds only the first occurrence of the specified L-profile.

Examples

```
L40*5 20
L50*50*5 25 | 25
L80*10 30 30 30 30 30 30
L200/15 40 45 40 40 40 35
L200/20 40 45 40 40 40 35
RSA45*45*5 20 25 20 20 20 20
RSA100*100*8 30 45 40 40 40 35
RSA75*150*15 30 35 30 30 30 25 | 35 40 35 35 35 30
RSA150*75*15 35 40 35 35 35 30 | 30 35 30 30 30 25
RSA200*200*20 40 45 40 40 40 35
```

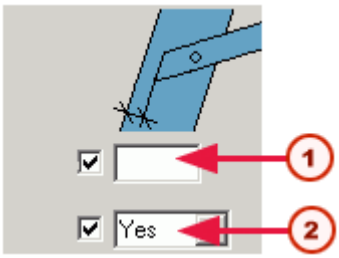
Moving and cutting braces (S67)

To move the ends of braces:



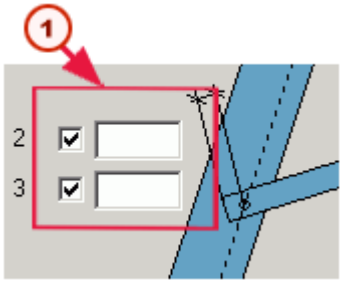
	Description
1	Define the bolt gage lines for the braces
2	Define the bolt gage line for the tower leg
3	Move points
4	Indicate which brace(s) to move
5	Enter the distance to move the point(s) where the gage lines intersect

To cut the braces:



	Description
1	Enter the distance from the end of the brace(s) to the edge of the tower leg
2	Select **Yes** to cut the brace(s)

To move bolt groups:



	Description
1	Enter the distance from the first bolt in the group to the end of the brace

Defining bolt properties

This section explains how to define the properties of bolts in tower components.

Click the links below to find out more:

- About bolt gage lines (page 3233)
- Editing default gage lines (page 3235)
- Creating bolts (87) (page 3236)
- Creating bolts (89) (page 3236)
- Creating bolts (178) (page 3237)
- Creating bolts (181) (page 3237)
- Creating bolts (182) (page 3237)
- Bolt location (87, 89) (page 3237)

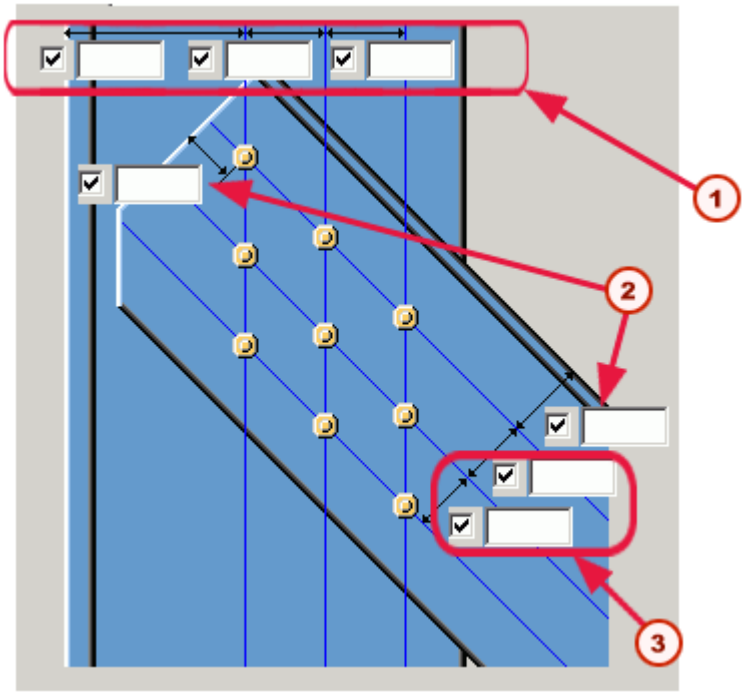
About bolt gage lines

Several components use gage lines that you can use to:

- Define the location of bolts on a brace
- Adjust the position of individual bolts
- Remove bolts

For example, **Leg 1 - diagonal (178)** uses gage lines.

Gage lines specify several dimensions:



	Description
1	Distance between bolts, horizontally
2	Distance from the center of the bolt to the edge of the brace
3	Distance between bolts, vertically

To use a default set of gage lines:

- Ensure that the `gauge_lines.dat` file is in the profile folder of the environment your are using,
- Leave all the fields blank on the **Parameters** tab.

TIP To learn how to change the default gage lines, see Editing default gage lines (page 3235).

Components using gauge_lines.dat

The following components use the `gauge_lines.dat` file:

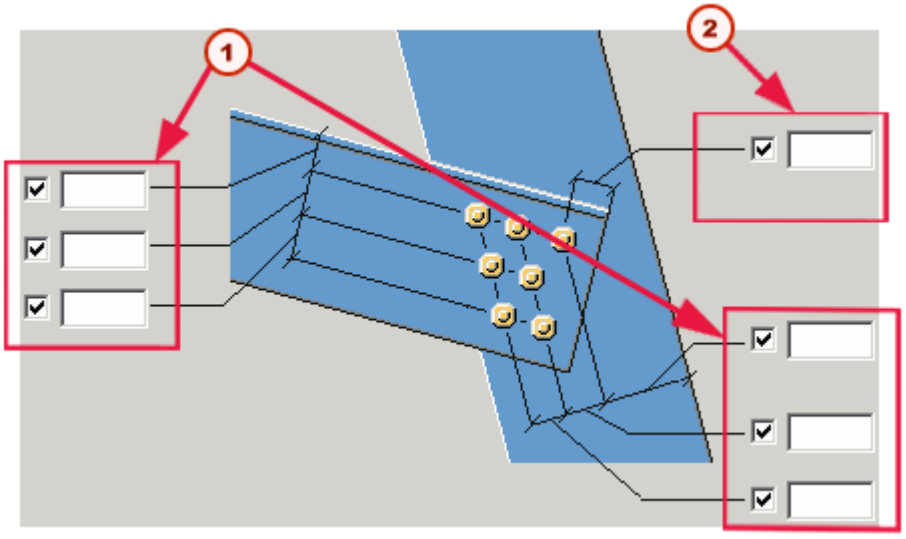
- **Tower Generation Macro (S43)**
- **Tower diagonal (S66)**
- **Auto position (S67)**
- **Batten plates (S85)**
- **Windbrace connection (110)**
- **Bent gusset (140)**
- **L splice (175)**
- **Parallel L profiles (176)**
- **Leg - 1 Diagonal (178)**

See also

Bolt location (87, 89) (page 3237)

Defining bolt gage lines (87)

To define bolt gage lines for **Tower 1 diagonal (87)**, go to the **Picture** tab and enter the following dimensions:



	Description
1	Location of gage lines
2	Location of gage lines

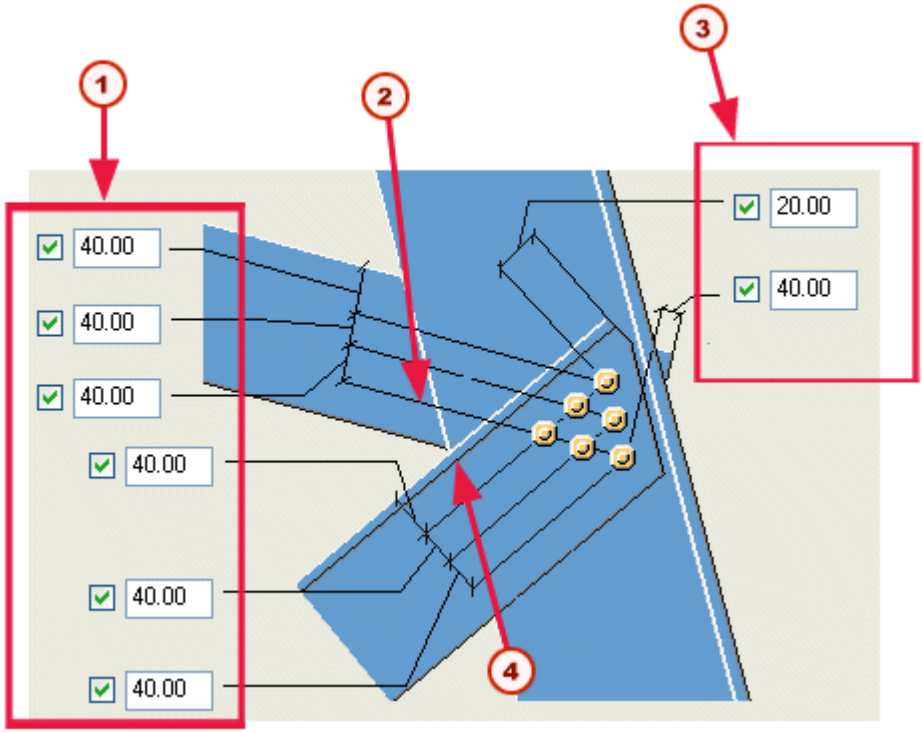
To learn how to create bolts, see Bolt location (87, 89) (page 3237).

Defining bolt gage lines (89)

To define bolt gage lines for each bolt group in **Tower 2 diagonal (89)**, go to the following tabs:

Bolt group	Tab
Connects all parts	**Picture**
Connects first and second brace picked to the tower leg	**Picture 2**

For example, on the **Picture** tab, enter the following dimensions:



	Description
1	Location of gage lines
2	Gage line 1(second brace picked)
3	Bolt edge distances
4	Gage line 1 (first brace picked)

To learn how to create bolts, see Bolt location (87, 89) (page 3237).

Editing default gage lines

To change the default gage lines for all components that use them, edit the file `gauge_lines. dat`, with any text editor (e.g. Notepad). The file is located in the system folder.

Creating bolts (87)

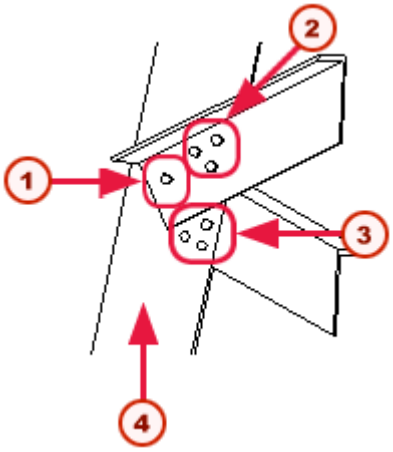
This component creates a single bolt group that connects the brace to the tower leg. To create bolts, you need to:

- Define the bolt gage lines and edge distances. See About bolt gage lines (page 3233)
- Create bolts, and specify the location of individual bolts. See Bolt location (87, 89) (page 3237)

WARNING This component does not create any bolts by default. So you must define the bolts to use.

Creating bolts (89)

This component creates several bolt groups:



	Description
1	Bolt group that connects all parts
2	Bolt group that connects the first brace picked to the tower leg
3	Bolt group that connects the second brace picked to the tower leg. (component 89 only)
4	Tower leg

For each bolt group, you need to:

- Define the bolt gage lines and edge distances. See About bolt gage lines (page 3233)
- Create bolts, and specify the location of individual bolts. See Bolt location (87, 89) (page 3237)

WARNING This component does not create any bolts by default. So you must define the bolts to use.

Creating bolts (178)

To create bolts, go to the **Parameters** tab and define the bolt gage lines. See About bolt gage lines (page 3233).

WARNING This component does not create any bolts by default, so you must define the bolts to use.

Creating bolts (181)

By default, this component creates one bolt at the intersection of the main bolt gages of the horizontal brace and the diagonal brace. To create a second bolt:

1. Go to the **Picture** tab and define the main and secondary bolt gages for the braces:
2. Go to the **Parameters** tab. Select one of the options in the **Switch for common bolt** dropdown list. The default option is **None**.

Creating bolts (182)

By default, Tekla Structures creates the following bolts:

- One bolt where the main bolt gages of the braces intersect, which connects all the braces to the plate.

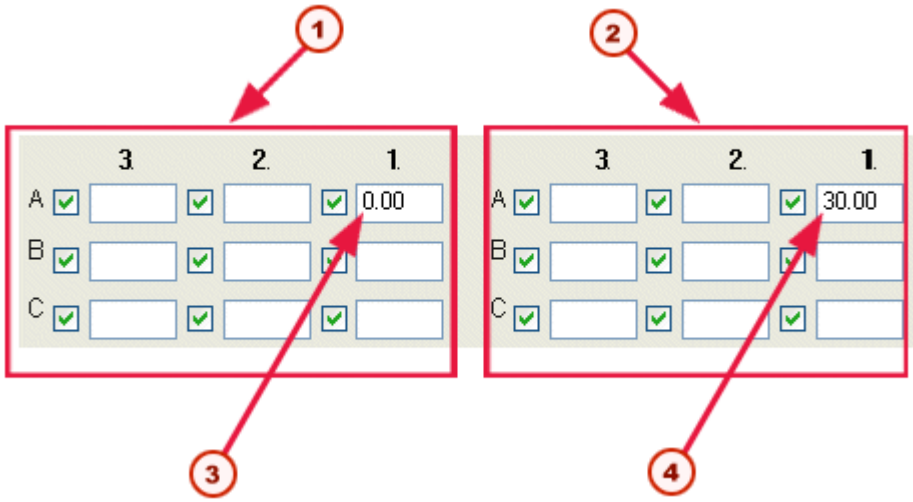
  To create a second bolt, go to the **Parameters** tab. Select one of the options in the **Switch for common bolt** dropdown list. The default option is **None**.
- A bolt on the main bolt gage line of each brace, which connects the brace to the plate.

Bolt location (87, 89)

Use the fields at the bottom of the **Picture** tab pages to create bolts in each bolt group, use the fields at the bottom of the **Picture** tab page(s):

Bolt group	Tab
Connects all parts	**Picture**
Connects first and second brace picked to the tower leg	**Picture 2**

- Enter 0 to create a bolt on the intersection of gage lines
- Enter 1 or more to move the bolt along the gage line, away from the end of the brace, on either the first or second diagonal brace picked:



	Description
1	First brace picked
2	Second brace picked (component 89 only)
3	Creates bolt at intersection of gage lines
4	Moves bolt 30 mm along the gage line, away from the end of the brace

TIP To move a bolt towards the edge of the brace, enter a negative number, e.g. -10.

Defining connection material

This section explains how to define the properties of connection material in tower components.

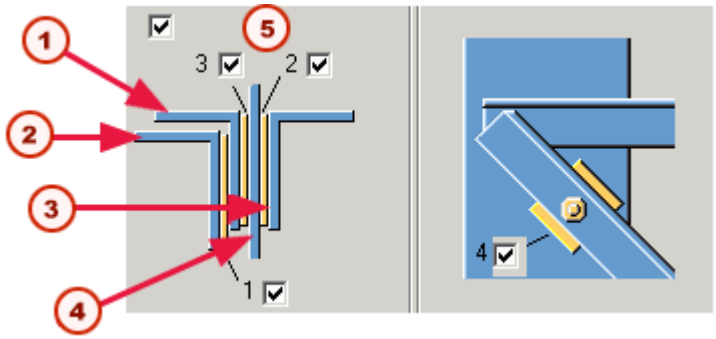
Click the links below to find out more:

- Defining filler plates (177) (page 3239)

- Defining filler plates (182) (page 3240)

Defining filler plates (177)

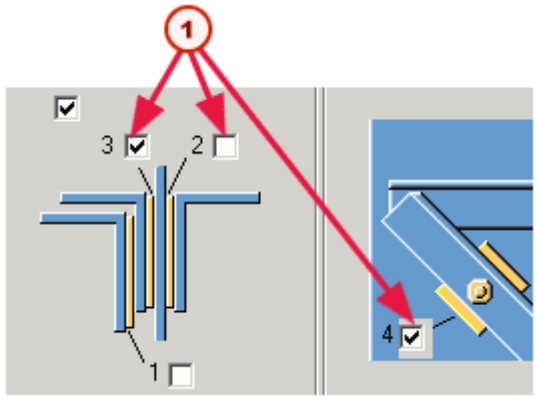
If necessary, **Leg - 2 & 3 diagonals (177)** automatically creates filler plates in the gaps between the braces and the tower leg.



	Description
	Plate 1: Between the first diagonal brace picked and the horizontal brace
	Plate 2: Between the second diagonal brace picked and the tower leg
	Plate 3: Between the horizontal brace and the tower leg
	Plate 4: Between the first horizontal brace picked and the tower leg
1	Horizontal brace
2	First diagonal brace picked
3	Second diagonal brace picked
4	Tower leg
5	Effect in modify

To specify the properties of each filler plate, use the fields on the **Plates** tab.

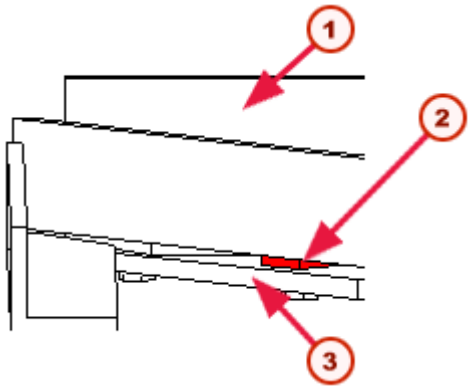
To remove a filler plate, go to the illustration and clear the checkbox against it:



	Description
1	To remove a plate, clear its checkbox. Here we are removing plates 1 and 2

Defining filler plates (182)

If the diagonal brace connects to the inside of the horizontal brace, Tekla Structures creates one or more filler plates to fill the gap between the diagonal brace and the plate:



	Description
1	Diagonal brace
2	Filler plate
3	Plate

To replace the filler plate with a ring or square washer, go to the **Parameters** tab and select an option in the **Switch to manage what kind of filler plate** list box.

5.19 Connection Map

This section introduces example illustrations of different connections, grouped according to their usage.

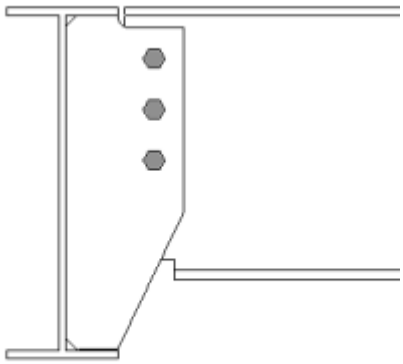
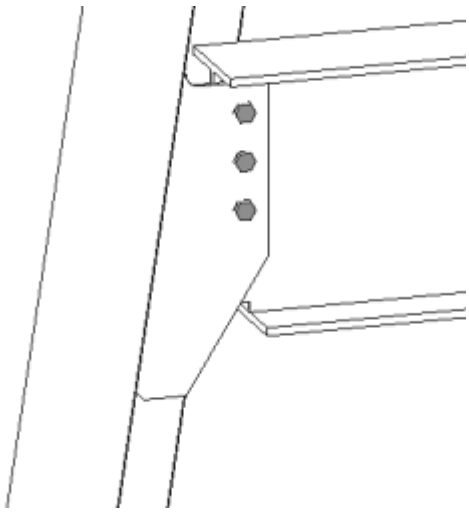
Click the links below to find out more:

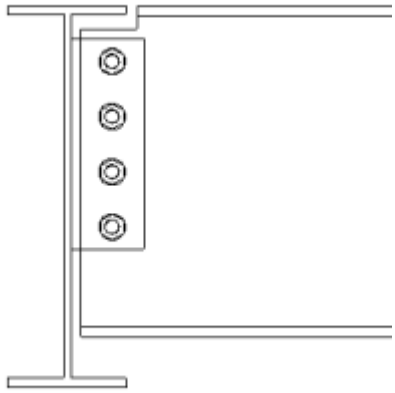
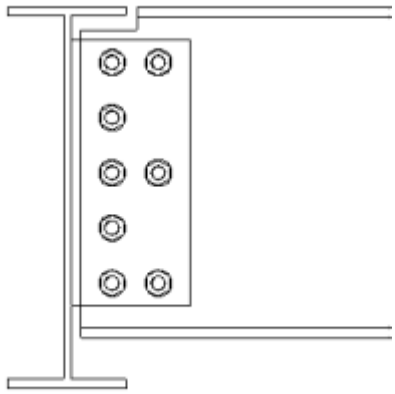
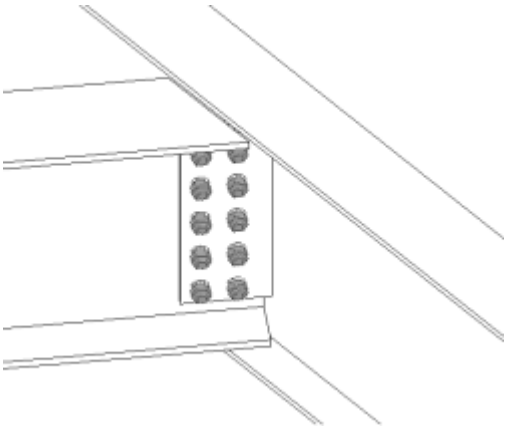
- Beam to beam framing connections (page 3241)
- Beam to column framing connections (page 3259)
- Splice connections (page 3281)
- Joist connections (page 3288)
- Vertical member to beam (page 3291)

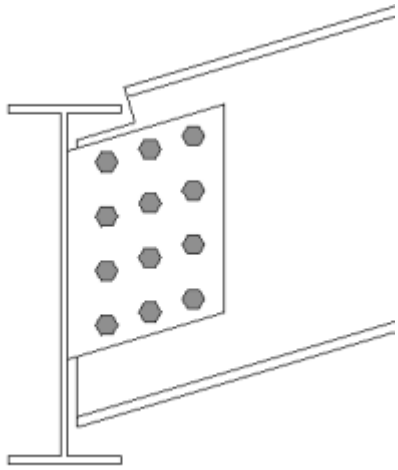
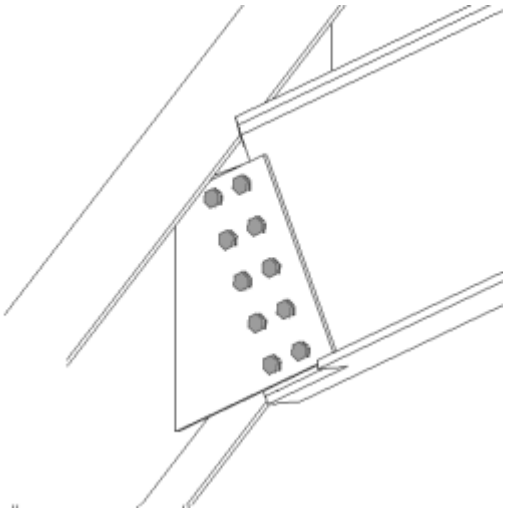
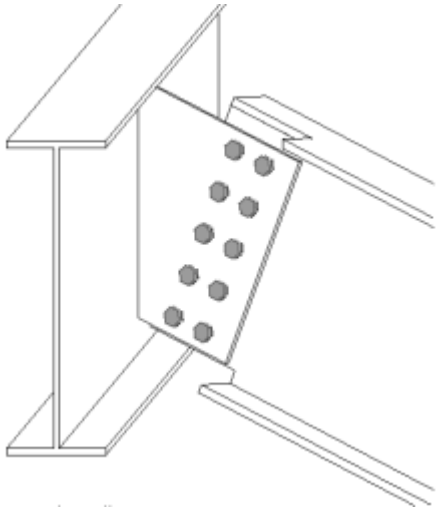
- Bracing connections (page 3295)
- Welded connections (page 3303)
- Details (page 3307)

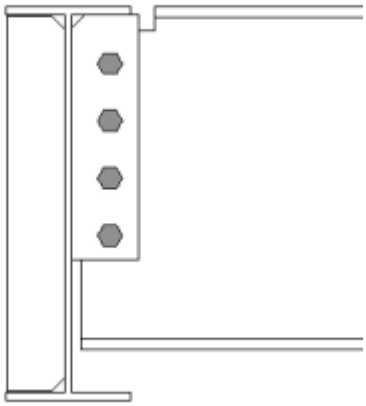
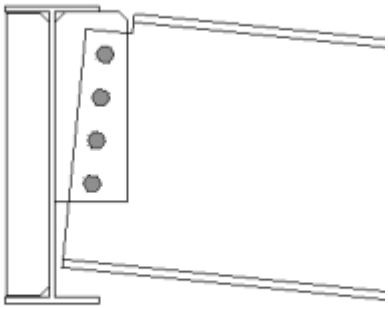
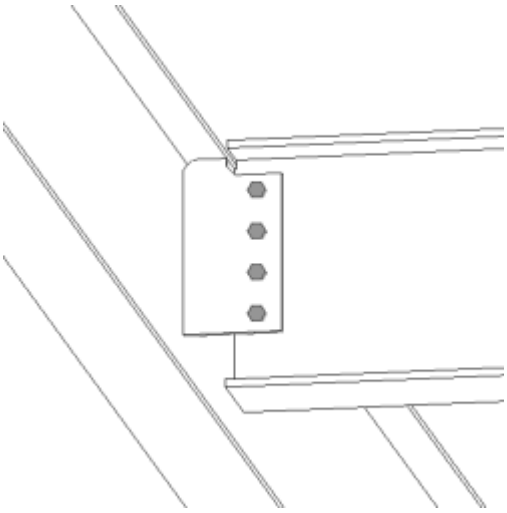
Beam to beam framing connections

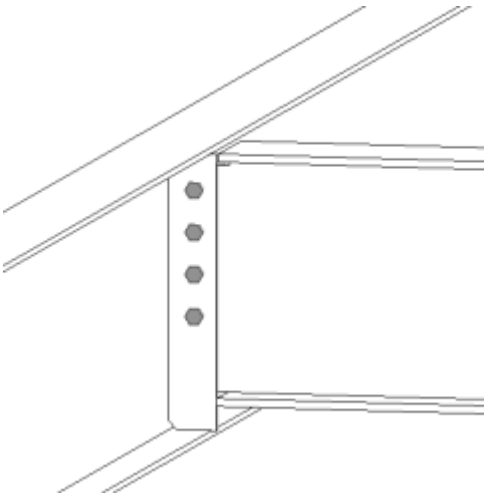
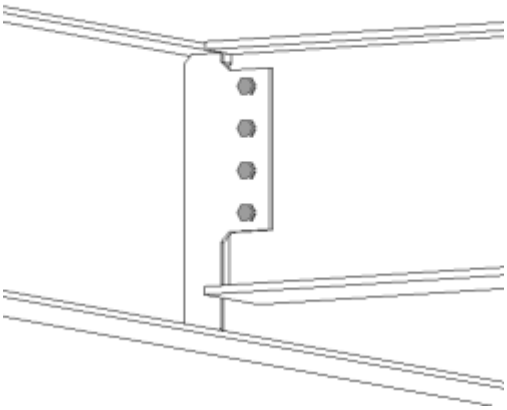
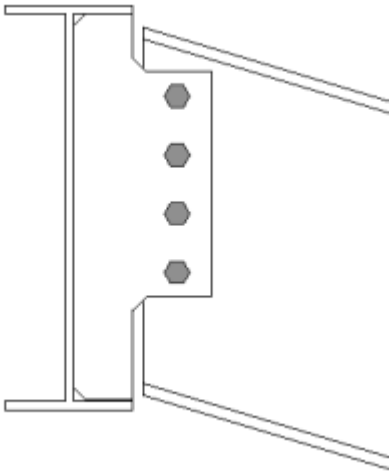
Shear tabs

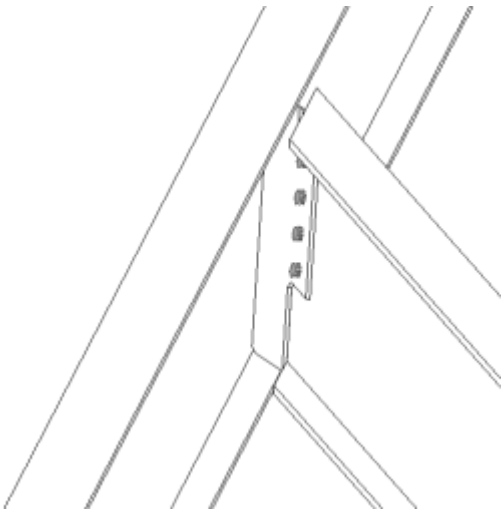
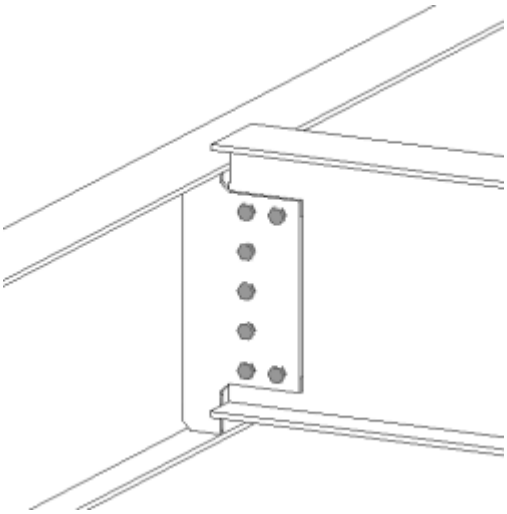
Example	Description
	Full depth shear tab - secondary beam cut short of main part. Use **Beam with stiffener (129)**.
	Full depth shear tab - secondary part sloped and/or skewed. Use **Beam with stiffener (129)**.

Example	Description
	Simple shear tab to beam. Use **Shear plate simple (146)**.
	Simple shear tab to beam - bolt elimination option. Use **Shear plate simple (146)**.
	Simple shear tab to beam - skewed secondary part. Use **Shear plate simple (146)**.

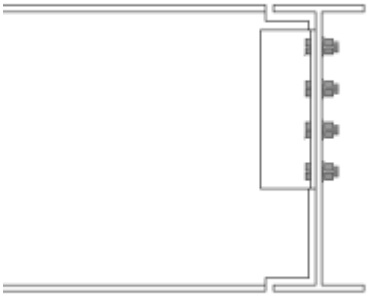
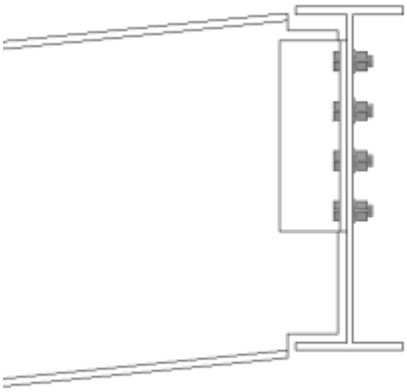
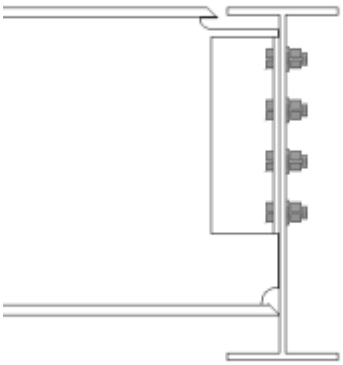
Example	Description
	Simple shear tab to beam - sloped (and skewed) secondary part. Bolts and plate oriented with secondary part. Use **Shear plate simple (146)**.
	Simple shear tab to beam - sloped and skewed secondary part. Valley condition. Use **Shear plate simple (146)**.
	Simple shear tab to beam - sloped and skewed secondary part. Valley condition. Use **Shear plate simple (146)**.

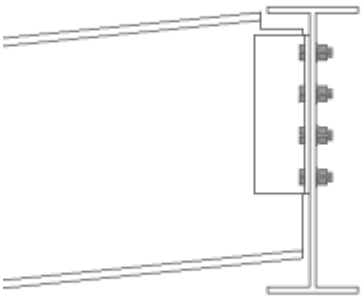
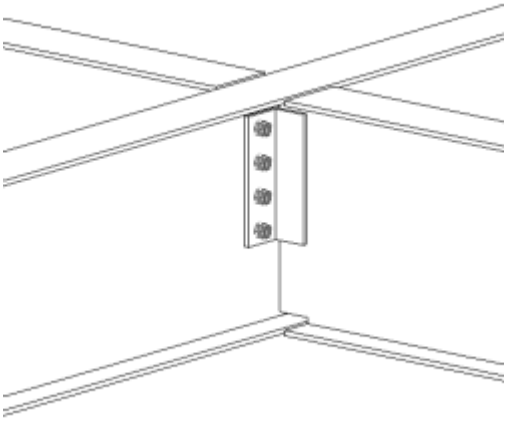
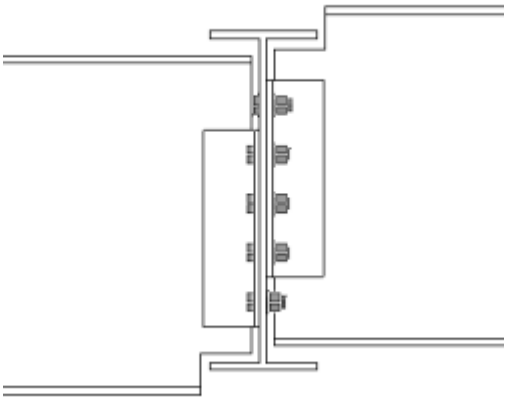
Example	Description
	Partial depth shear tab to top flange of beam - square or skewed, stiffener option. Use **Welded to top flange (147)**.
	Partial depth shear tab to top flange of beam. Sloped/ square or skewed. Use **Welded to top flange (147)**.
	Partial depth shear tab to top flange of beam, secondary beam cut short of main part. Square, sloped, skewed. Use **Welded to top flange S (149)**.

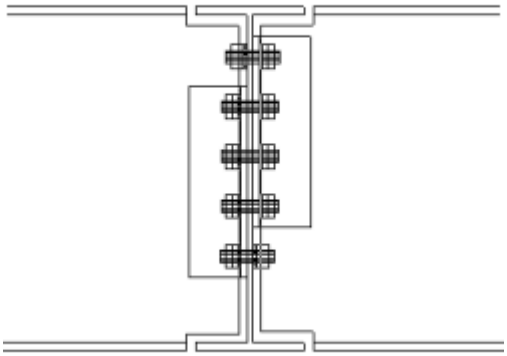
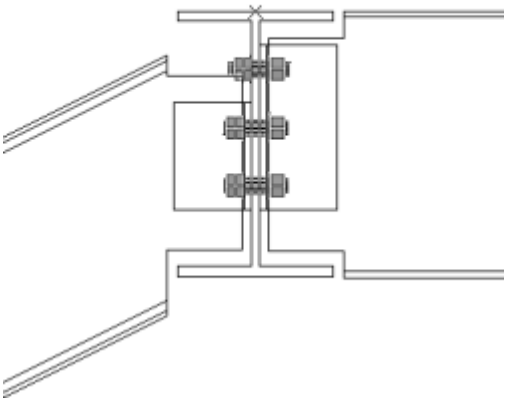
Example	Description
	Full depth shear tab. Square, sloped, skewed. Use **Full depth (184)**.
	Full depth shear tab. Secondary beam cut short of main part. Use **Full depth S (185)**.
	Full depth shear tab. Secondary beam cut short of main part. Sloped secondary. Use **Full depth S (185)**.

Example	Description
	Full depth shear tab. Secondary beam cut short of main part. Sloped and skewed secondary (hip & valley). Use **Full depth S (185)**.
	Full depth shear tab. Secondary beam cut short of main part. Secondary offset. Bolt elimination option. Use **Full depth S (185)**.

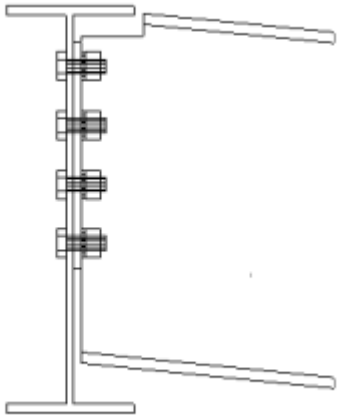
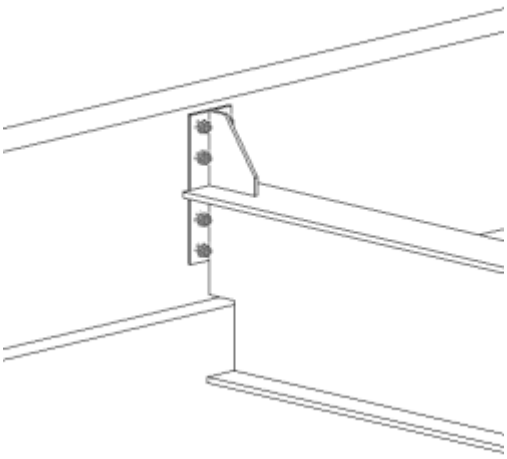
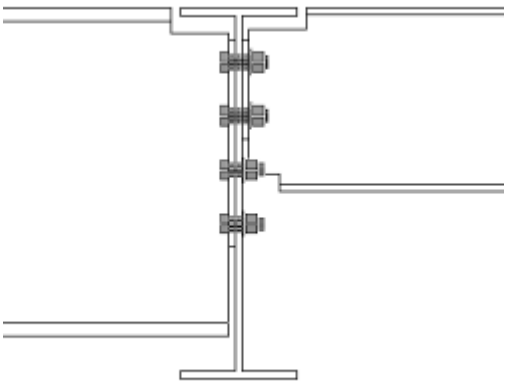
Clip angles

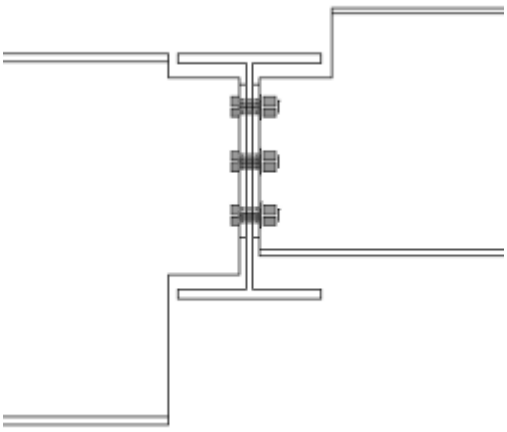
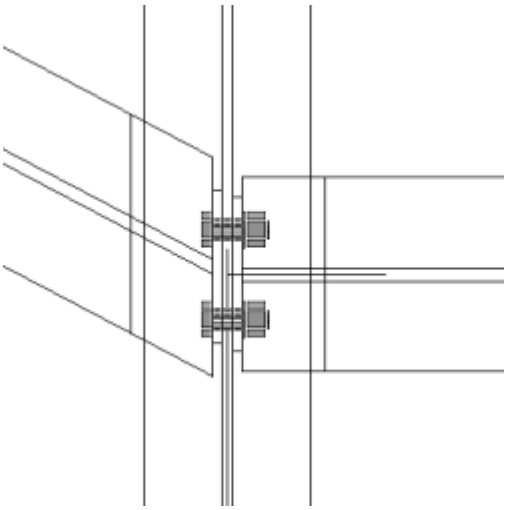
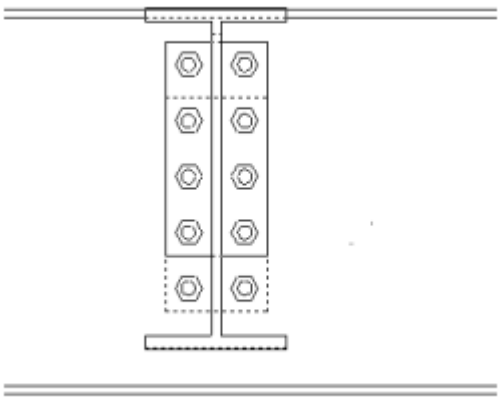
Example	Description
	Clip angle connection – single-sided clip/double-sided clip. Use **Clip angle (141)**.
	Clip angle connection – single-sided clip/double-sided clip. Sloped secondary part. Various notching options. Use **Clip angle (141)**.
	Clip angle connection – single-sided clip/double-sided clip. Weld preparation option. Use **Clip angle (141)**.

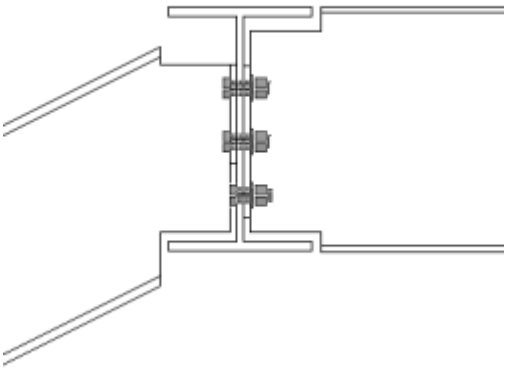
Example	Description
	Clip angle connection – single-sided clip/double-sided clip. Sloped secondary part. Use **Clip angle (141)**.
	Clip angle connection – single-sided clip/double-sided clip. Two secondary parts. Bolted/bolted, welded/bolted, welded/welded options. Use **Two sided clip angle (143)**.
	Clip angle connection – single-sided clip/double-sided clip. Two secondary parts at differing heights. Use **Two sided clip angle (143)**.

Example	Description
	Clip angle connection – single-sided clip/double-sided clip. Two secondary parts. Safety connection. Use **Two sided clip angle (143)**.
	Clip angle connection – single-sided clip/double-sided clip. Two secondary parts. One sloped. Use **Two sided clip angle (143)**.

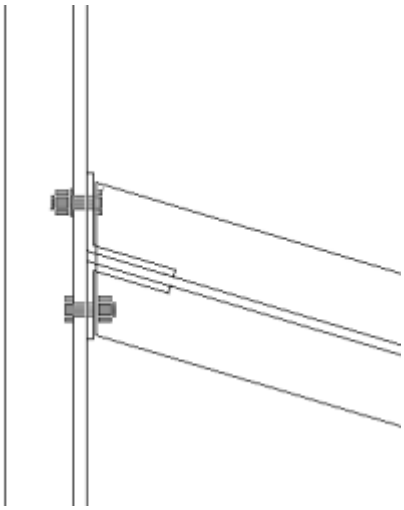
End plates

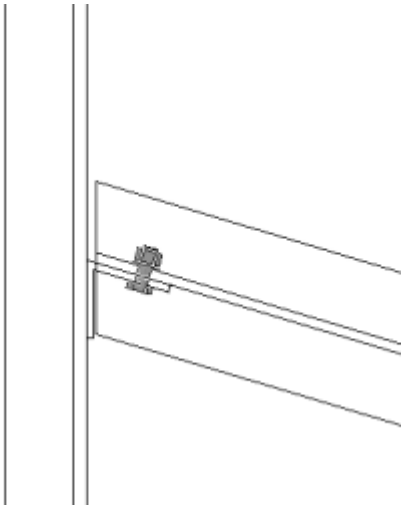
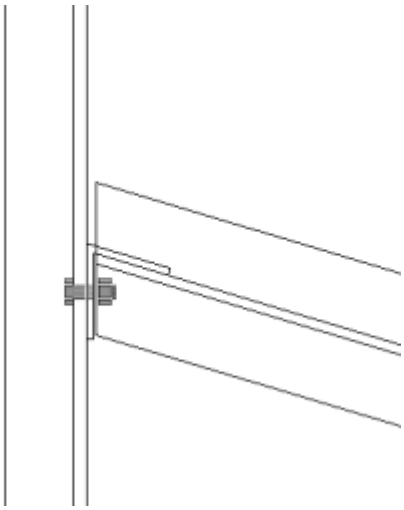
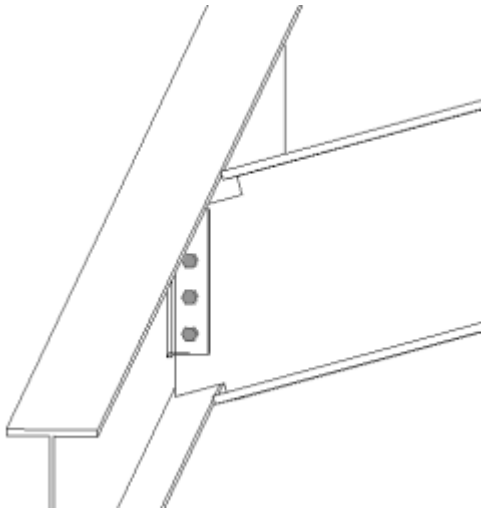
Example	Description
	End plate connection - level or sloped, square or skewed secondary part. Various notching options. Use **End plate (144)**.
	End plate connection – extended plate with or without haunches. Use **End plate (144)**.
	End plate connection – two secondary parts. Automatic notch for bolt clearance. Use **Two sided end plate (142)**.

Example	Description
	End plate connection – two secondary parts at differing heights. Use **Two sided end plate (142)**.
	End plate connection – two secondary parts. Square and /or skewed. Use **Two sided end plate (142)**.
	End plate connection – two secondary parts. Safety connection. Use **Two sided end plate (142)**.

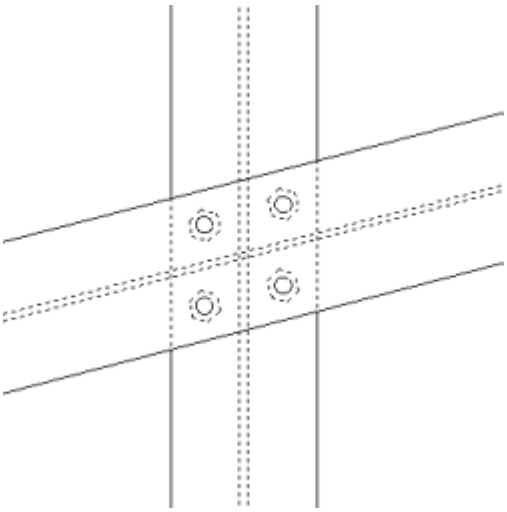
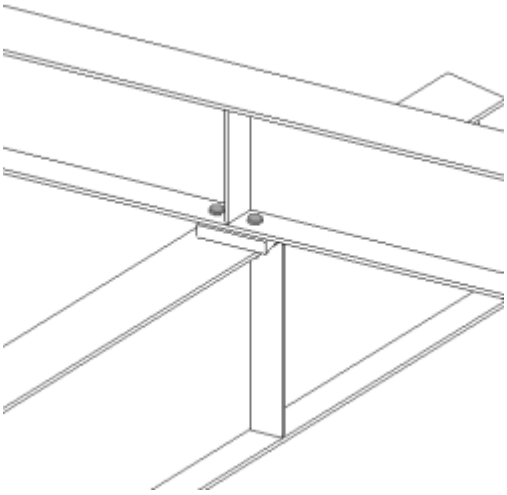
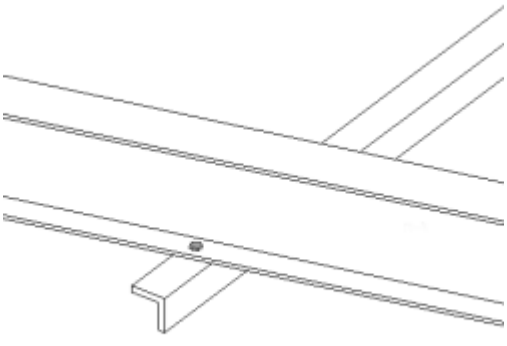
Example	Description
	End plate connection – two secondary parts. Level and/or sloped. Use **Two sided end plate (142)**.

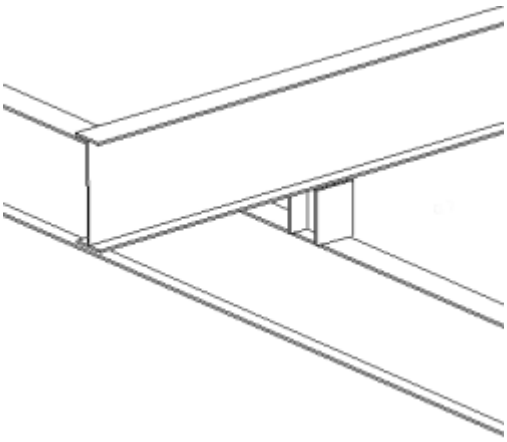
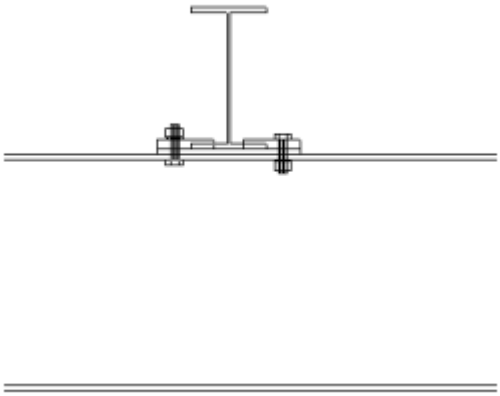
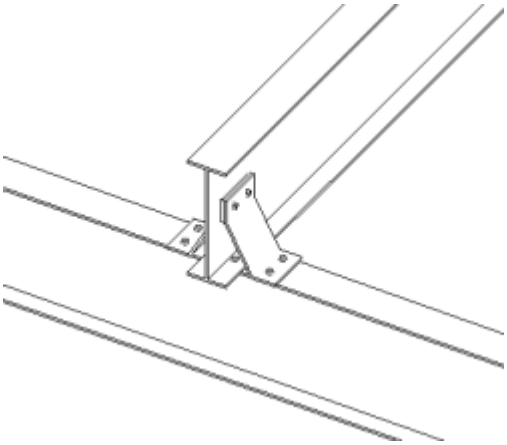
Bent plate

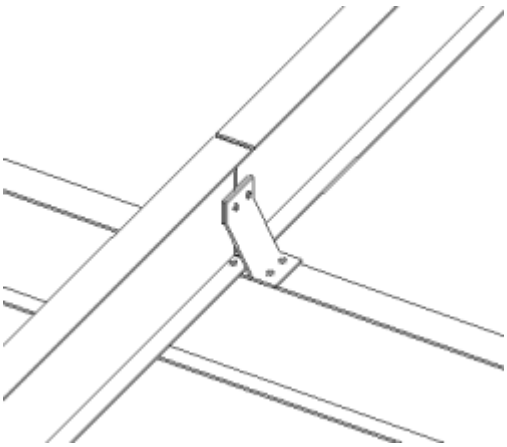
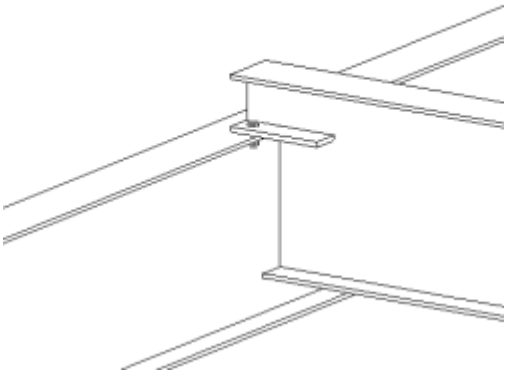
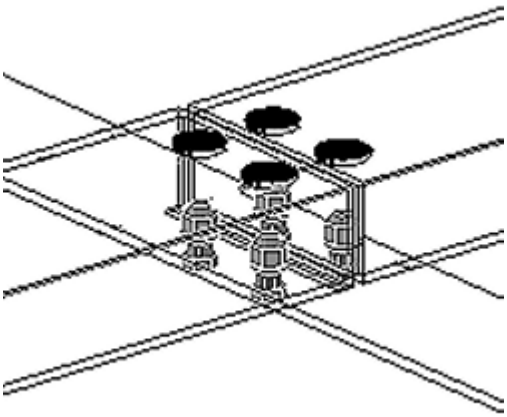
Example	Description
	Bent plate connection – skewed or square secondary part, plate near side and far side. Use **Bent plate (190)**.

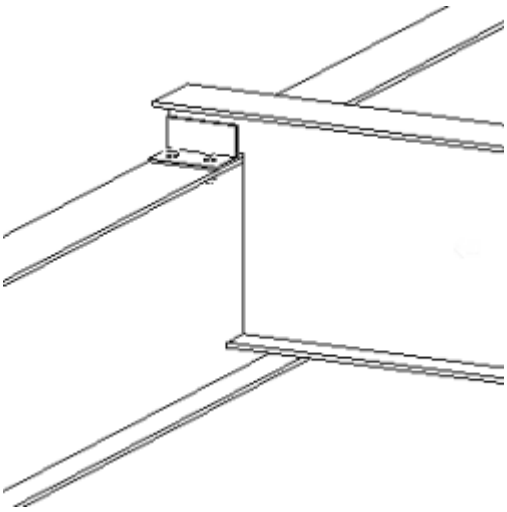
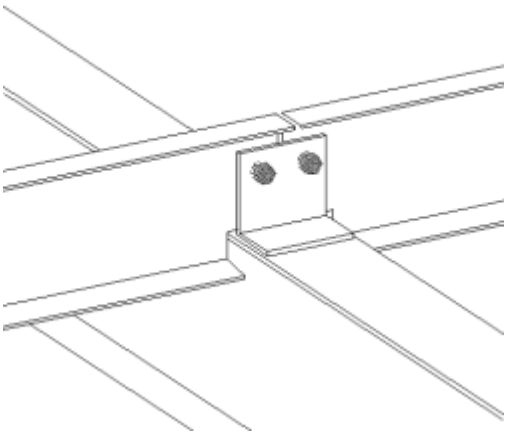
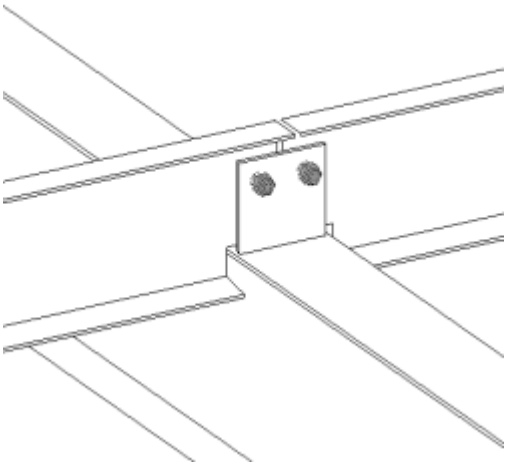
Example	Description
	Bent plate connection – skewed or square secondary part, plate one side. Use **Bent plate (190)**.
	Bent plate connection – skewed or square secondary part, plate one side. Various plate placing options. Use **Bent plate (190)**.
	Bent plate connection – skewed and sloped (hip & valley). Use **Bent plate (190)**.

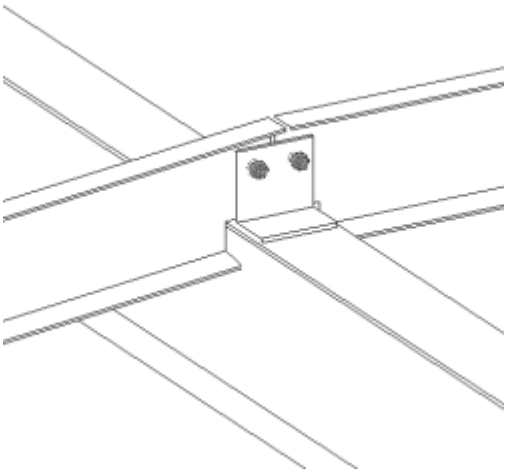
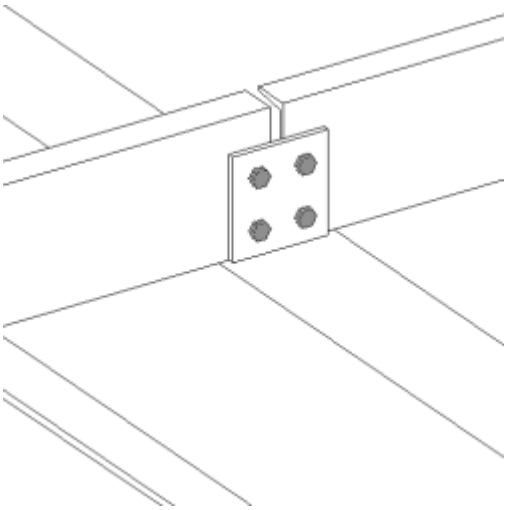
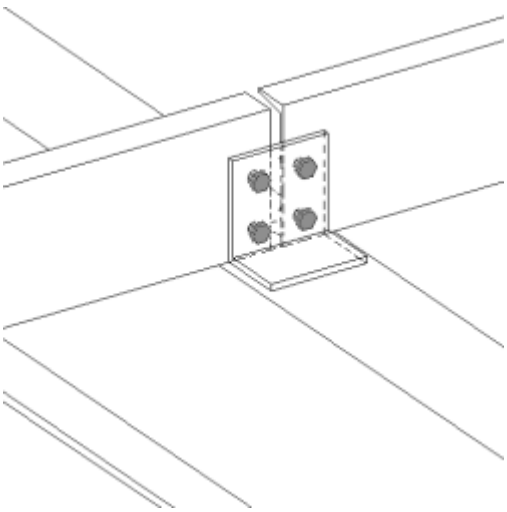
Bearing type

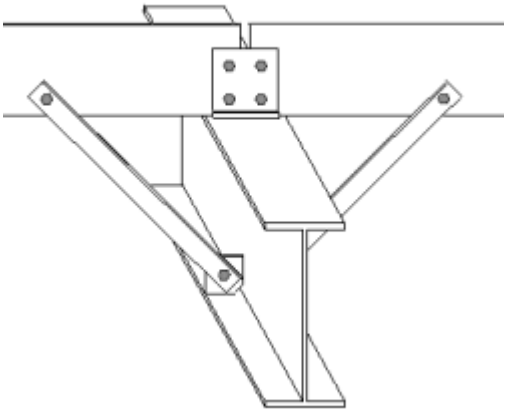
Example	Description
	Beam to beam bearing connection. Options for 1,2,3, or 4 bolts. Use **Seating (30)**.
	Beam to beam bearing connection. Spacer option. Use **Seating (30)**.
	Beam to beam bearing connection. Brace support. Use **Seating (30)**.

Example	Description
	Beam to beam bearing connection with stub column. Square, sloped and skewed conditions. Use **Cross (4)**.
	Beam to beam bearing clamp type connection. Use **Seating with nail (36)**.
	Beam to beam bearing purlin connection to single purlin. Use **Purlin connections (93)**.

Example	Description
	Beam to beam bearing purlin connection to two purlins. Use **Purlin connections (93)**.
	Beam to beam bearing notched seating connection. Square and skewed conditions. Use **Notched seating (9)**.
	Beam to beam bearing tube steel rails to main part. Cope or hole bolt access options. Use **Tube rail (113)**.

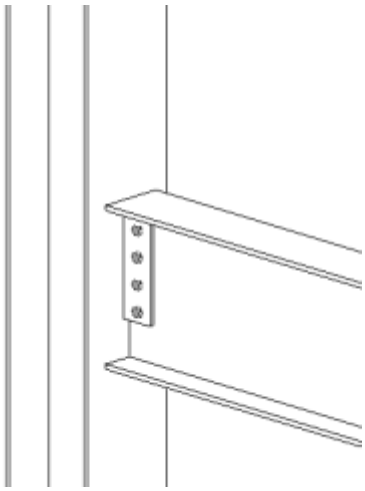
Example	Description
	Beam to beam bearing notched seat with angle. Use **Rail joint (70)**.
	Beam to beam bearing rafter connection to two secondary parts. Notching required. Use **Rail joint (70)**.
	Beam to beam bearing rafter connection to two secondary parts. Plate. Notching required. Use **Rail joint (70)**.

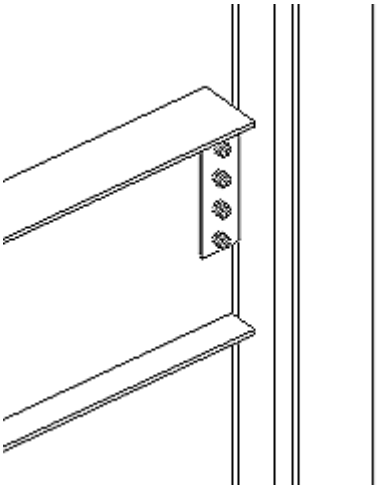
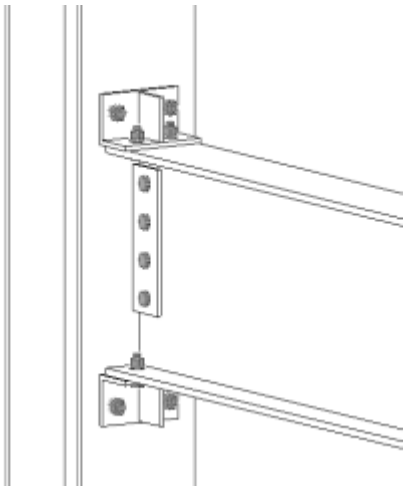
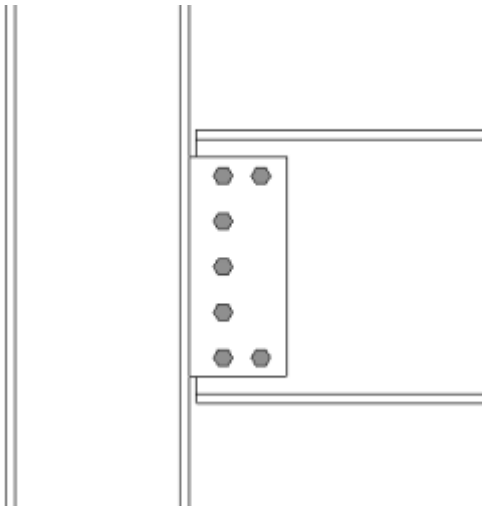
Example	Description
	Beam to beam bearing rafter connection to two secondary parts. Sloped/level notching required. Use **Rail joint (70)**.
	Beam to beam full bearing rafter connection to two secondary parts. Plate. Use **Cold rolled overlap (1)**.
	Beam to beam full bearing rafter connection to two secondary parts. Angle. Use **Cold rolled overlap (1)**.

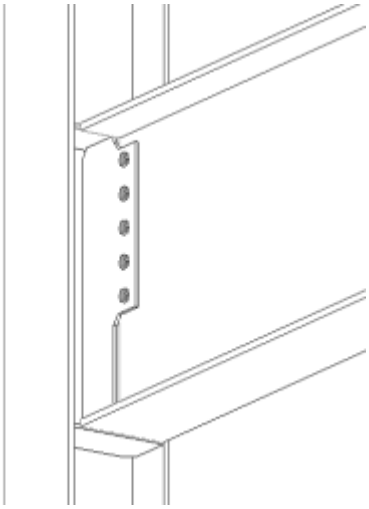
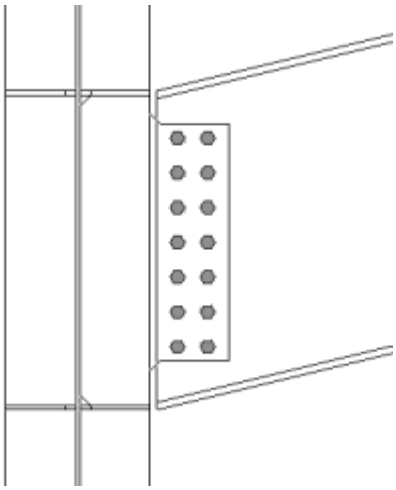
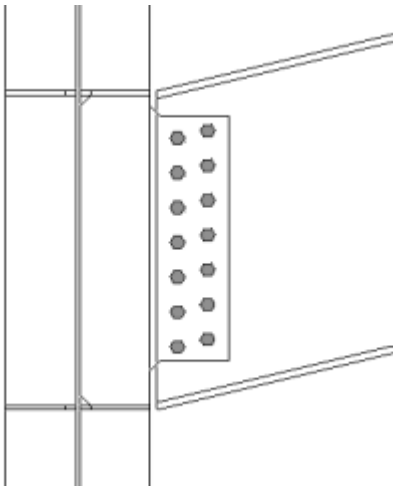
Example	Description
	Beam to beam full bearing rafter connection to two secondary parts. Knee brace option. Use **Cold rolled overlap (1)**.

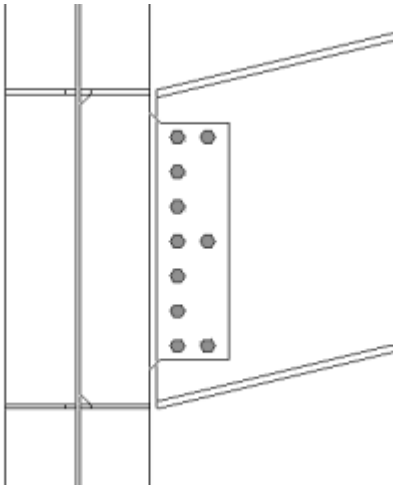
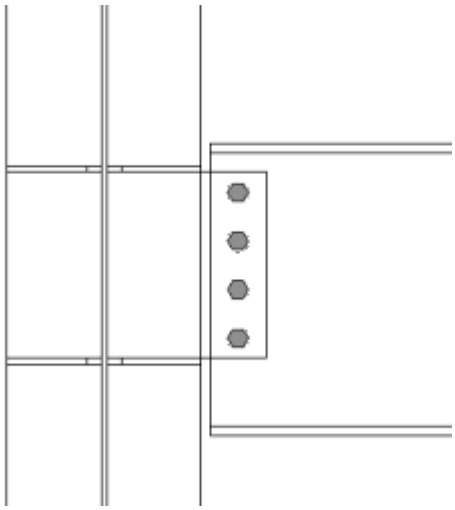
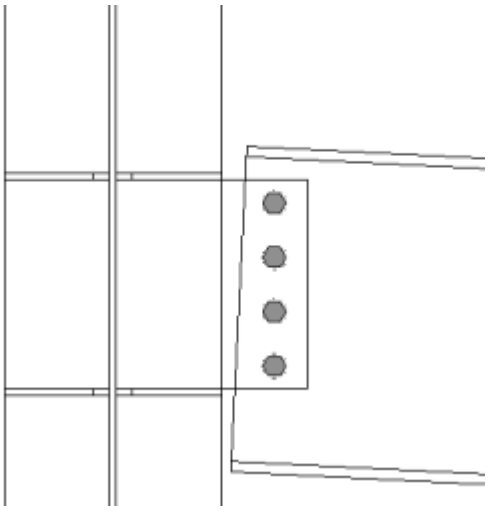
Beam to column framing connections

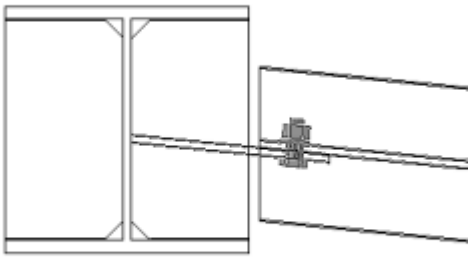
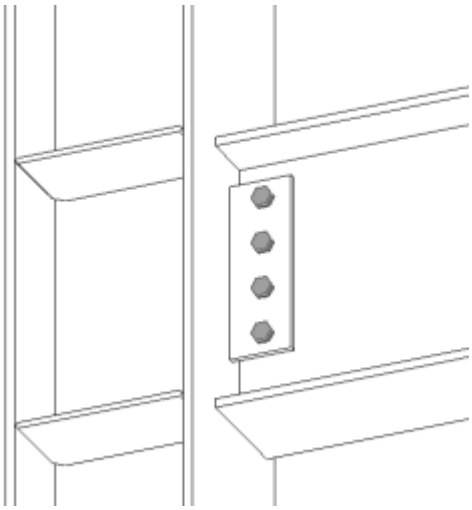
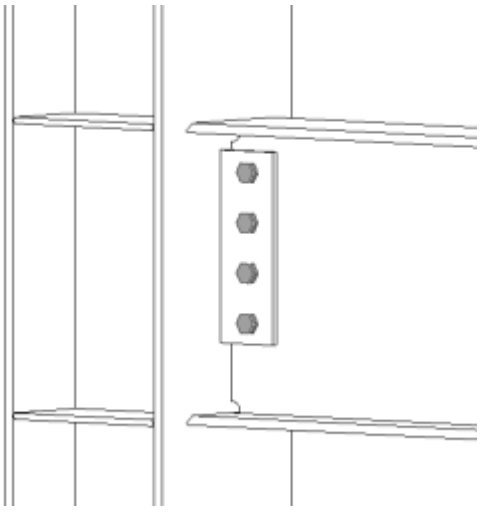
Shear tabs

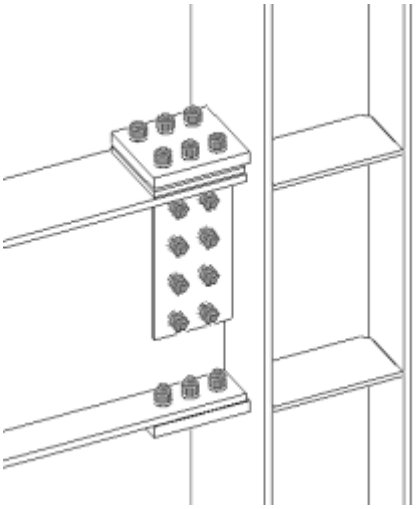
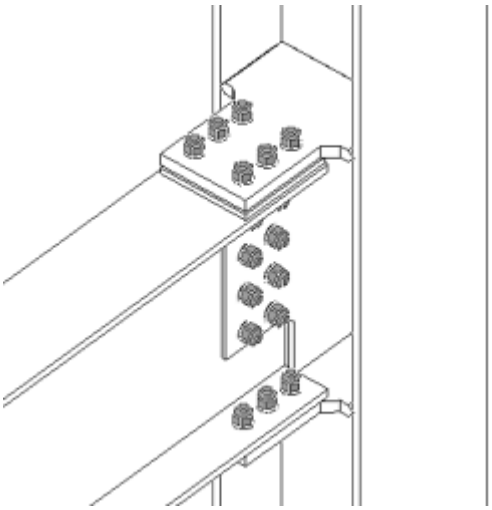
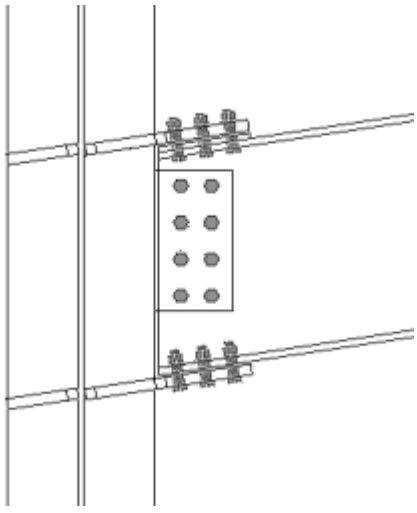
Example	Description
	Simple shear tab to column flange. Use **Shear plate simple (146)**.

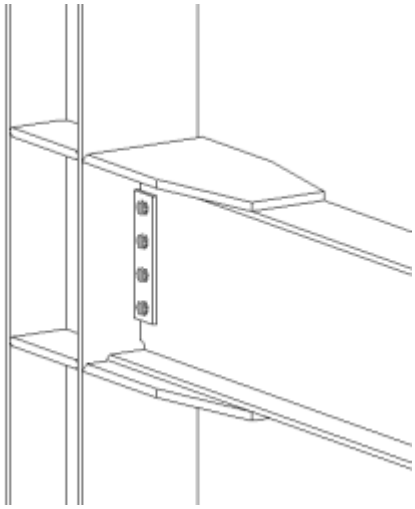
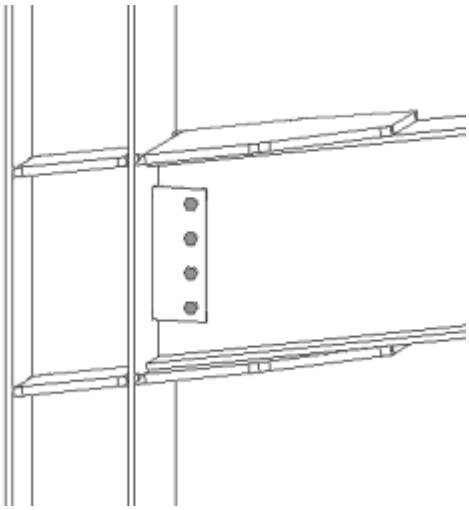
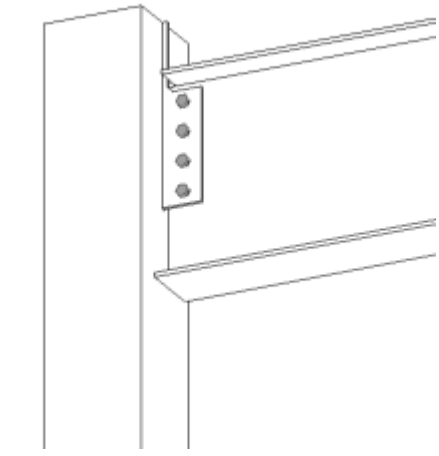
Example	Description
	Simple shear tab to edge/column flange. Use **Shear plate simple (146)**.
	Simple shear tab to column flange with seat angle options. Use **Shear plate simple (146)**.
	Simple shear tab to column flange. Bolt elimination options. Use **Shear plate simple (146)**.

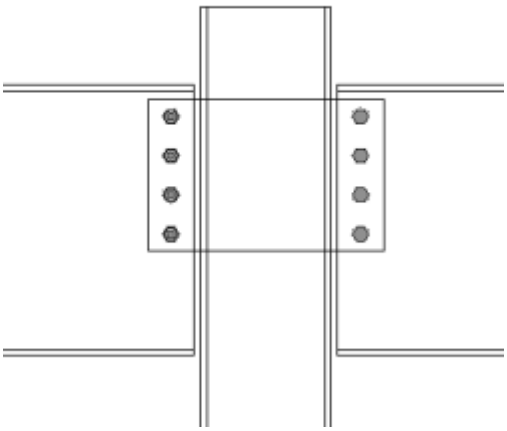
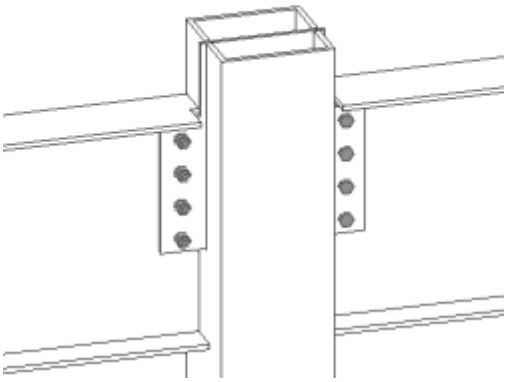
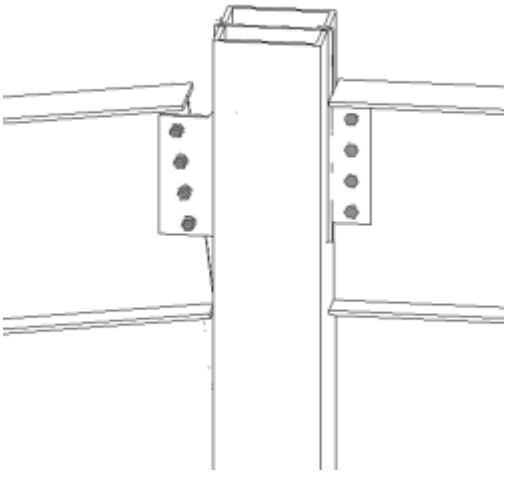
Example	Description
	Shaped shear tab with column stiffeners. Use **Column with stiffeners W (182)**.
	Shaped shear tab with column stiffeners. Sloped secondary. Use **Column with stiffeners W (182)**.
	Shaped shear tab with column stiffeners. Bolts aligned with secondary part. Use **Column with stiffeners W (182)**.

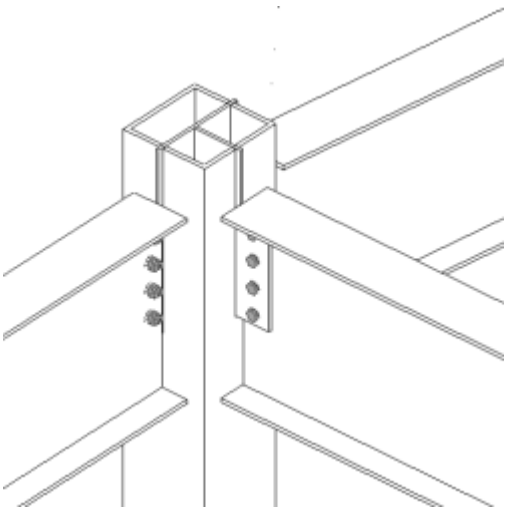
Example	Description
	Shaped shear tab with column stiffeners. Bolt elimination option. Use **Column with stiffeners W (182)**.
	Shear tab with column stiffeners. Use **Column with stiffeners (186)**.
	Shear tab with column stiffeners. Sloped secondary. Use **Column with stiffeners (186)**.

Example	Description
	Shear tab with column stiffeners. Skewed secondary. Use **Column with stiffeners (186)**.
	Shear tab to column flange with column stiffeners. Use **Column with stiffeners (188)**.
	Shear tab to column flange with column stiffeners. Weld preparation and weld access holes for moment connection option. Use **Column with stiffeners (188)**.

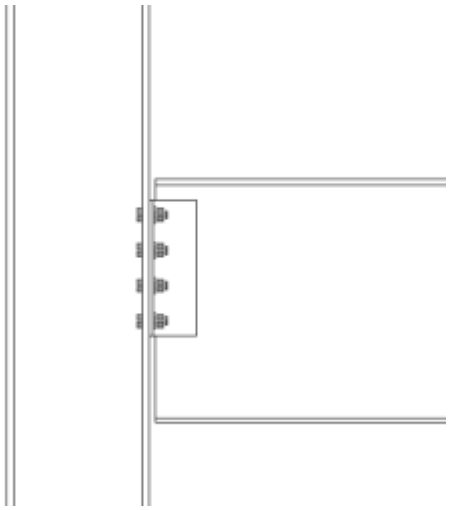
Example	Description
	Bolted moment connection to column flange with column stiffeners. Use **Bolted moment connection (134)**.
	Bolted moment connection to column web. Use **Bolted moment connection (134)**.
	Bolted moment connection to column web. Sloped secondary part. Use **Bolted moment connection (134)**.

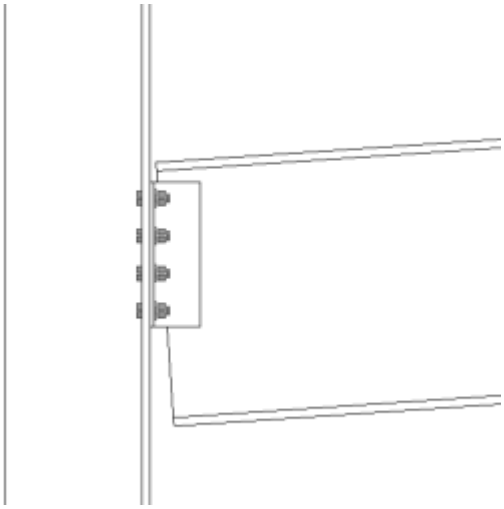
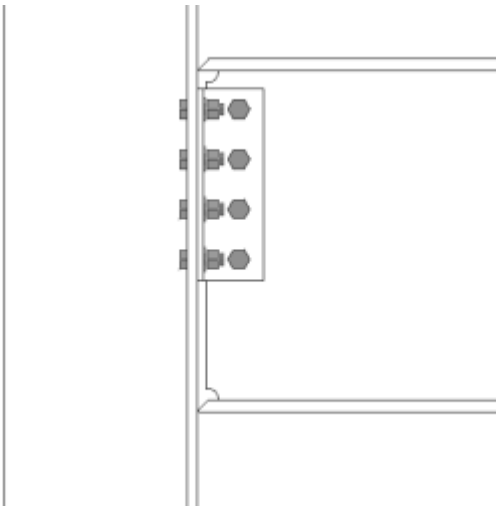
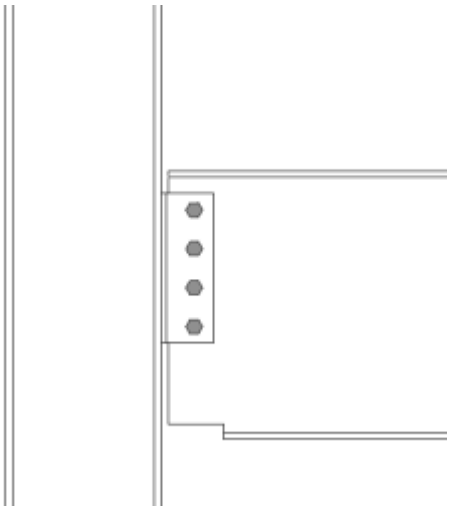
Example	Description
	Welded moment connection to column flange. Beam weld preparation and weld access hole options. Use **Moment connection (181)**.
	Welded moment connection to column flange. Sloped. Use **Moment connection (181)**.
	Shear tab through tube column. Use **Shear plate tube column (189)**.

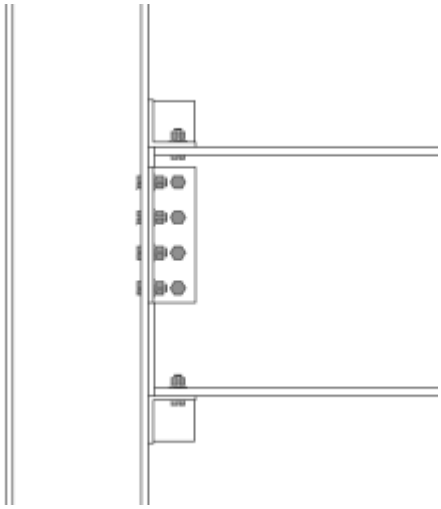
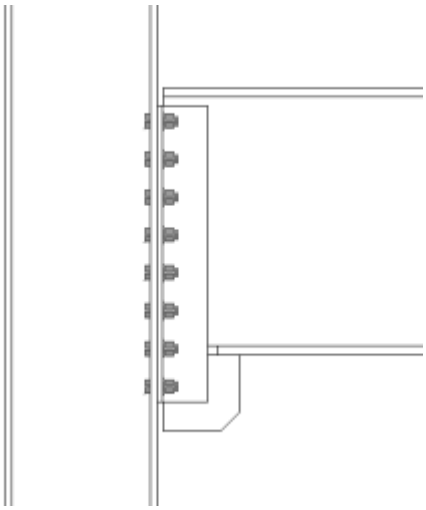
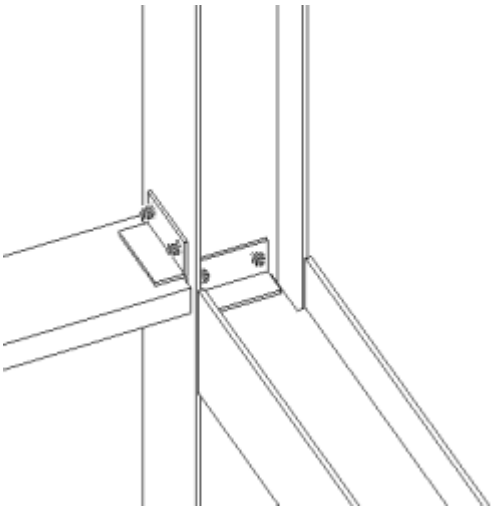
Example	Description
	Shear tab through tube column. Two secondary parts. Use **Shear plate tube column (189)**.
	Shear tab through tube column. Two secondary parts. Plate extended to top of column option. Use **Shear plate tube column (189)**.
	Shear tab through tube column. Two secondary parts, level and/or sloped. Bolt alignment options. Use **Shear plate tube column (189)**.

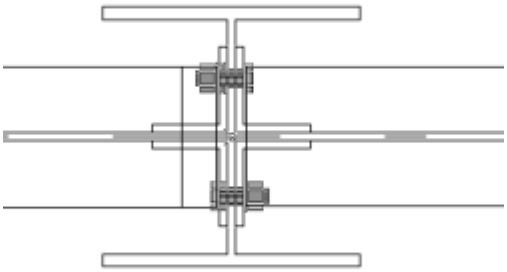
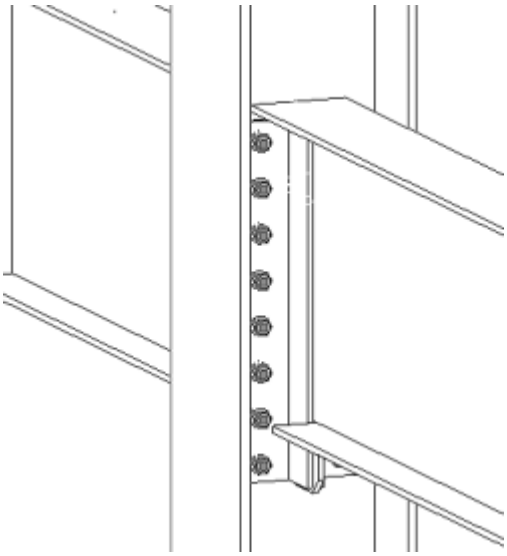
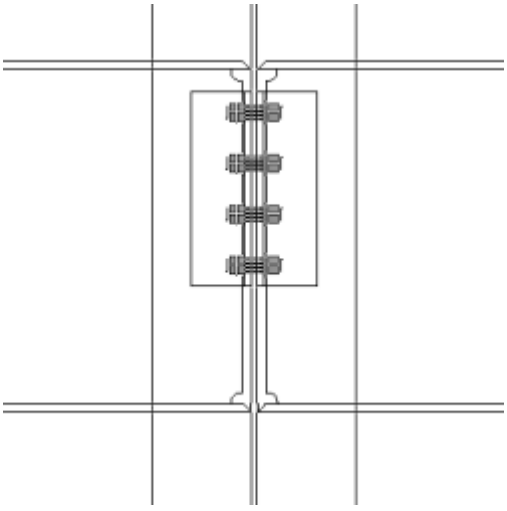
Example	Description
	Shear tab through tube column. Third secondary part after connection applied to original two secondary parts. Use **Shear plate tube column (189)**.

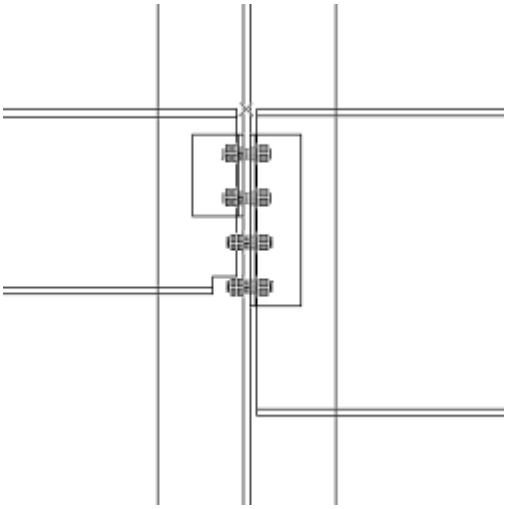
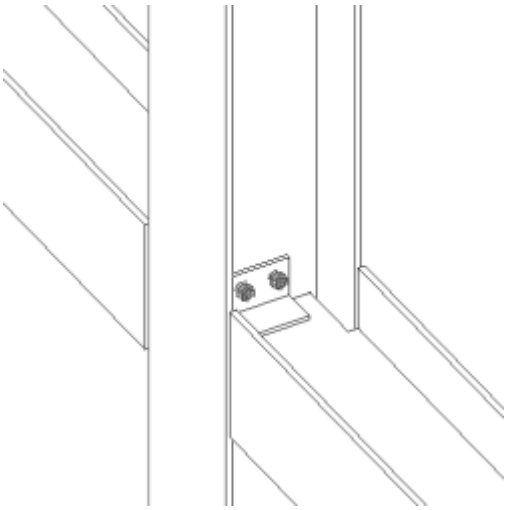
Clip angles

Example	Description
	Clip angle connection to column flange or web. Single-sided /double-sided clip. Welded/bolted, bolted/bolted, welded/welded options. Use **Clip angle (141)**.

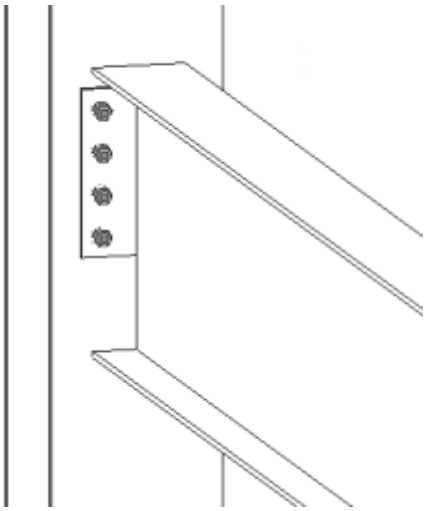
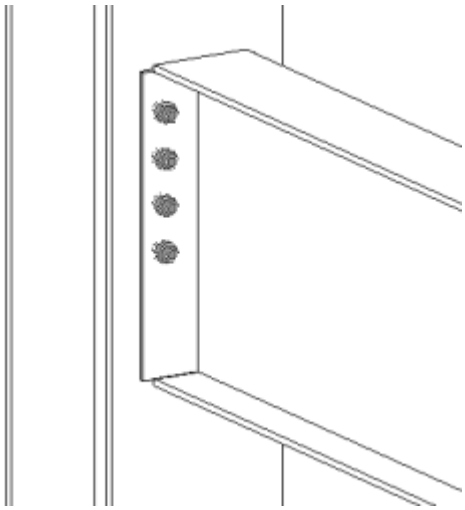
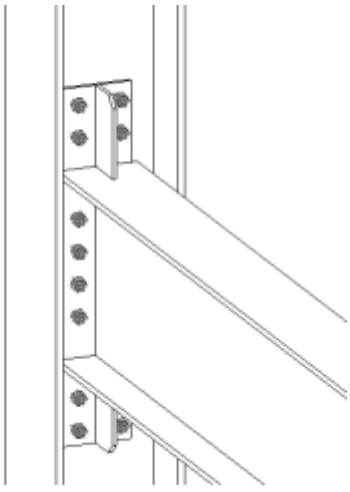
Example	Description
	Clip angle connection to column flange or web. Single-sided /double-sided clip. Sloped secondary part. Square or bevel cut secondary options. Use **Clip angle (141)**.
	Clip angle connection to column flange or web. Single-sided /double-sided clip. Weld preparation and weld access holes for moment connection. Use **Clip angle (141)**.
	Clip angle connection to column knife connection. Bottom flange blocked or stripped for erection. Use **Clip angle (141)**.

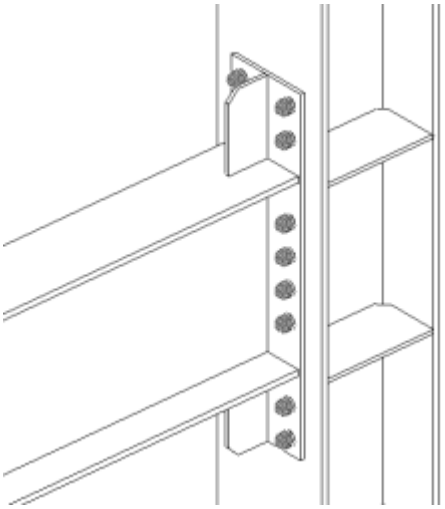
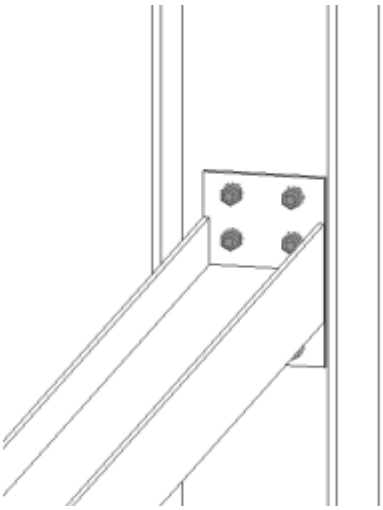
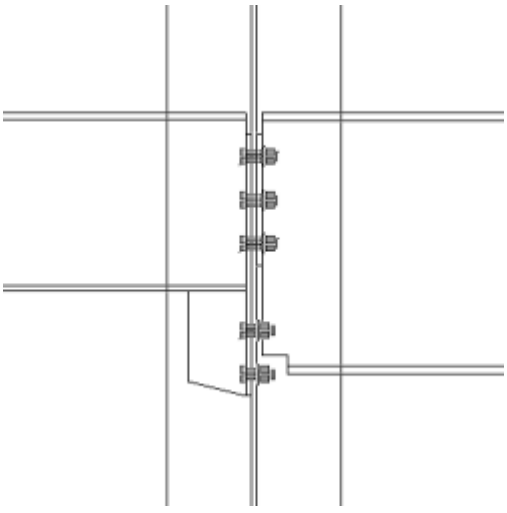
Example	Description
	Clip angle connection to column. Seat angle option. Top/Bottom/Both. Use **Clip angle (141)**.
	Clip angle connection to column flange or web. Single-sided /double-sided clip. Haunch extension option. Top/Bottom/Both. Use **Clip angle (141)**.
	Clip angle connection to column flange or web. Single-sided /double-sided clip. Rotated secondary part. Use **Clip angle (141)**.

Example	Description
	Clip angle connection. Single-sided / double-sided clip. Two secondary parts. Bolted/bolted, welded/bolted, welded/welded options. Use **Two sided clip angle (143)**.
	Clip angle connection. Single-sided / double-sided clip. Two secondary parts. Haunch extension option. Top/ Bottom/Both. Use **Two sided clip angle (143)**.
	Clip angle connection. Single-sided / double-sided clip. Two secondary parts. Weld preparation and weld access holes for moment connection. Use **Two sided clip angle (143)**.

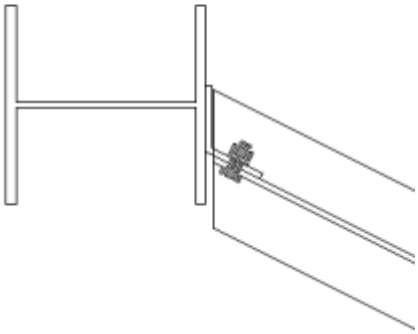
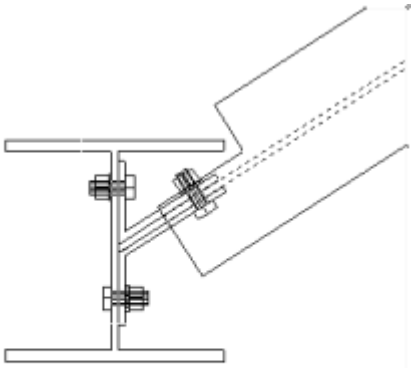
Example	Description
	Clip angle connection. Automatic notching of secondary part to provide bolt clearance. Use **Two sided clip angle (143)**.
	Clip angle connection. Single-sided / double-sided clip. Two secondary parts rotated. Use **Two sided clip angle (143)**.

End plates

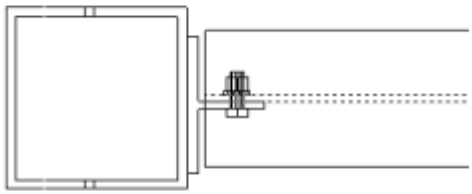
Example	Description
	End plate connection to column flange or web. Level or sloped, square or skewed secondary part. Use **End plate (144)**.
	Full depth end plate connection to column flange or web. Level or sloped, square or skewed secondary part. Use **End plate (144)**.
	End plate connection to column web. Extended plate with haunches option. Use **End plate (144)**.

Example	Description
	End plate connection to column flange. Column stiffener option. Use **End plate (144)**.
	End plate connection to column. Secondary part rotated. Use **End plate (144)**.
	End plate connection. Two secondary parts. Automatic notching for bolt clearance. Haunch option. Use **Two sided end plate (142)**.

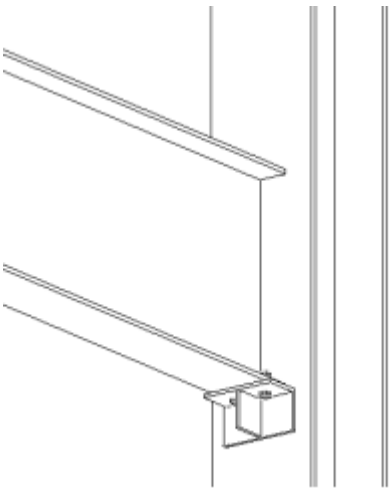
Bent plate

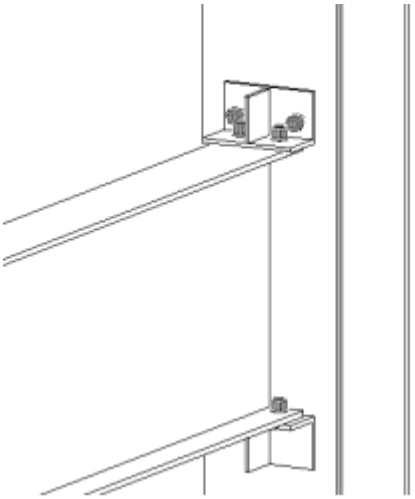
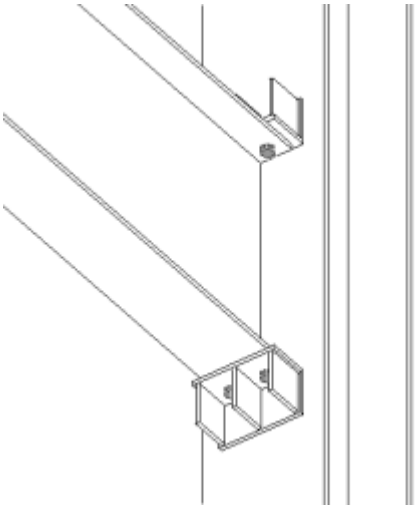
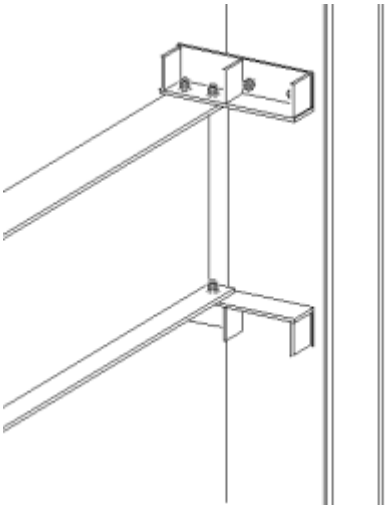
Example	Description
	Bent plate connection to column flange. Skewed or square secondary part. Plate near side\far side\both sides. Use **Bent plate (190)**.
	Bent plate connection to column web. Skewed or square secondary part. Plate near side\far side\both sides. Use **Bent plate (190)**.

Welded tee

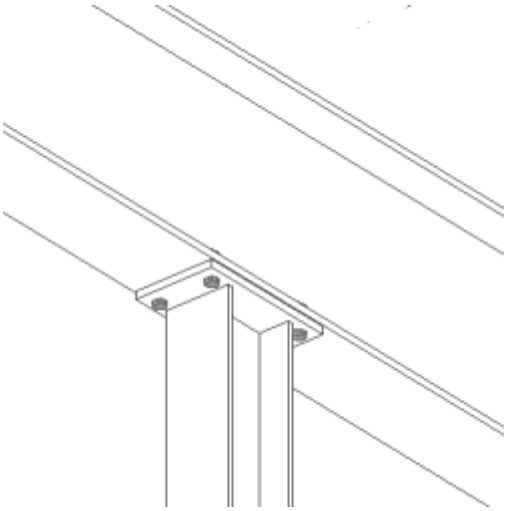
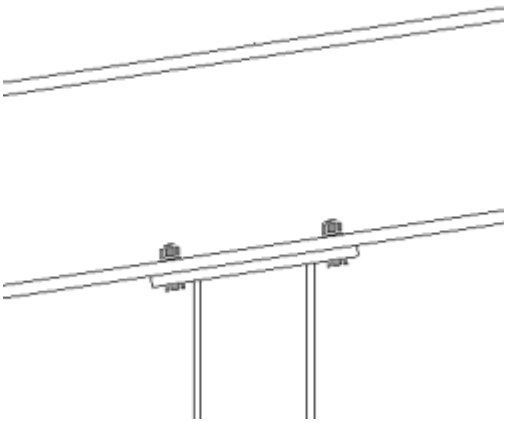
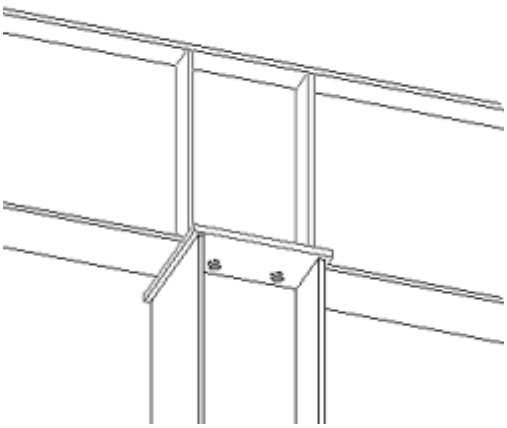
Example	Description
	Welded tee to column. Use **Welded tee (32)**.

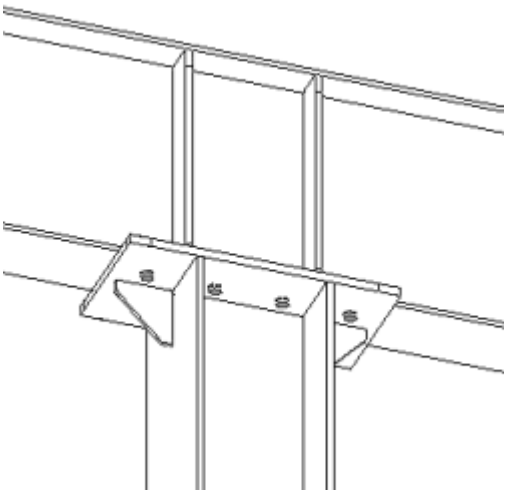
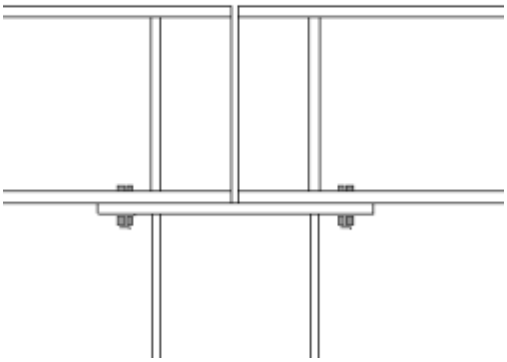
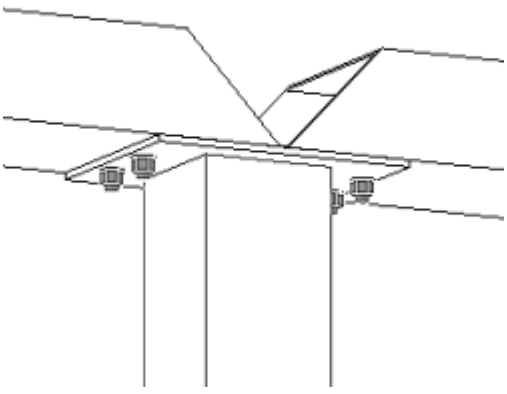
Seated connection

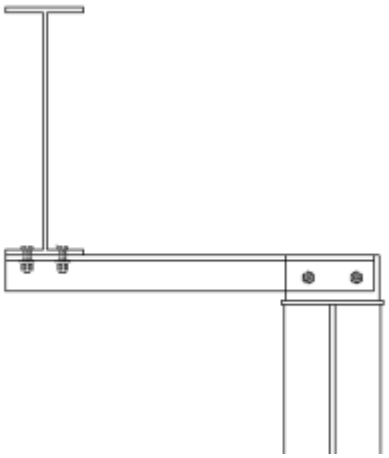
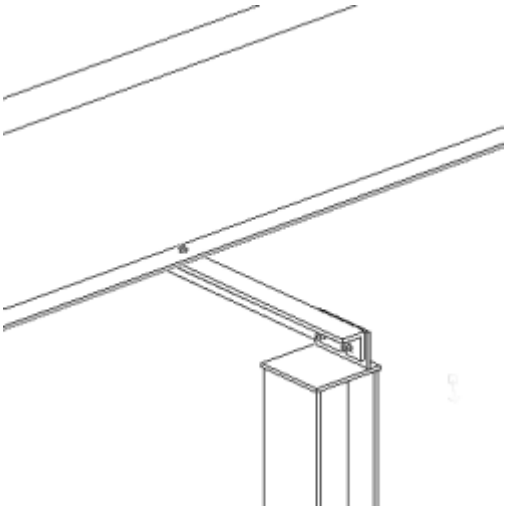
Example	Description
	Beam seat with stiffeners. Use **Rail joint (170)**.

Example	Description
	Beam seat top and bottom with stiffeners. Various bolting options. Use **Rail joint (170)**.
	Beam seat. Multiple stiffener options. Use **Rail joint (170)**.
	Beam seat. Offset secondary part. Use **Rail joint (170)**.

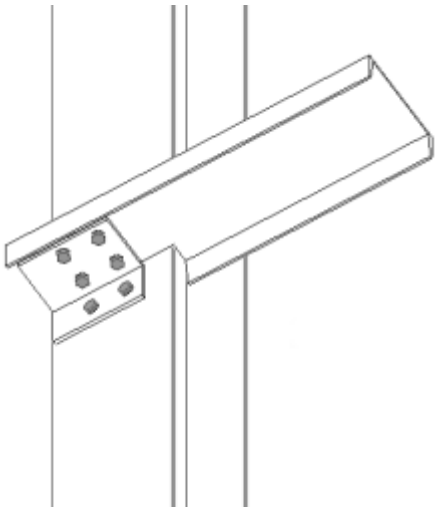
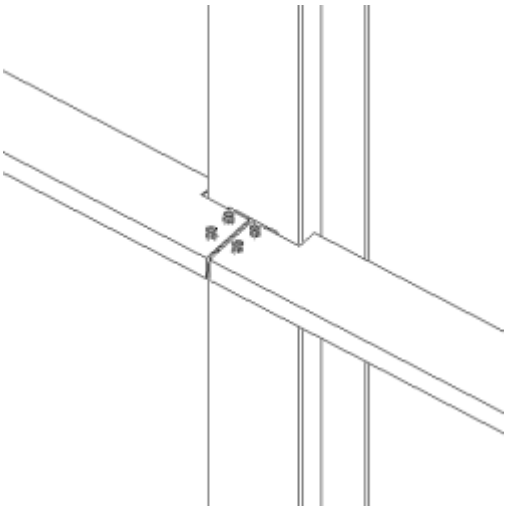
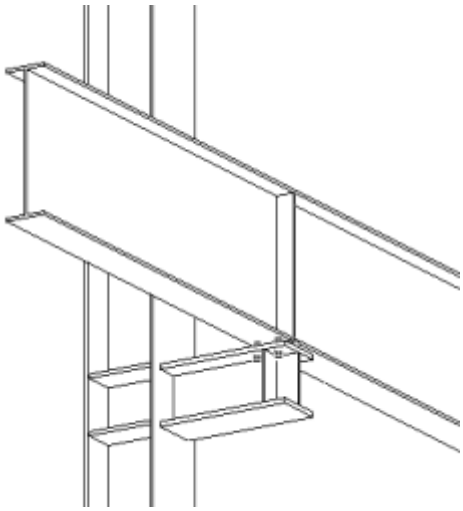
Bearing type cap plate

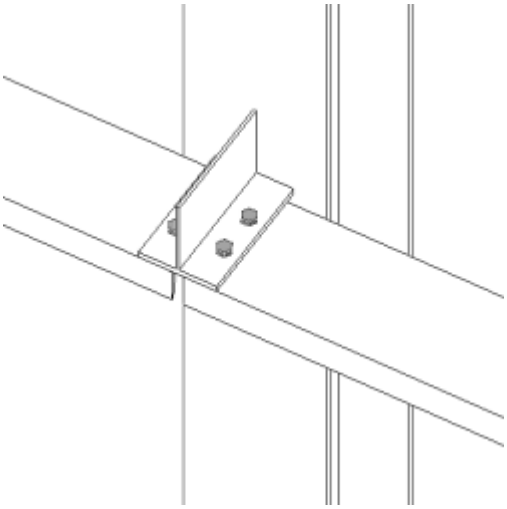
Example	Description
	Beam cantilever over column with cap plate. Use **U.S. Base plate joint (71)**.
	Beam cantilever over column with cap plate. Sloped secondary part. Use **U.S. Base plate joint (71)**.
	Beam cantilever over column with cap plate. Beam stiffener option. Use **U.S. Base plate joint (71)**.

Example	Description
	Beam cantilever over column with cap plate. Column stiffener option. Use **U.S. Base plate joint (71)**.
	Two beams to column cap plate. Beam stiffener options. Use **Seating (39)**.
	Beam column cap plate. Tube steel rails to main part. Cope or hole bolt access options. Use **Column tube seating (100)**.

Example	Description
	Wind column to beam connection. Use **Wind column (5)**.
	Wind column to beam connection. Plate offset option. Use **Wind column (5)**.

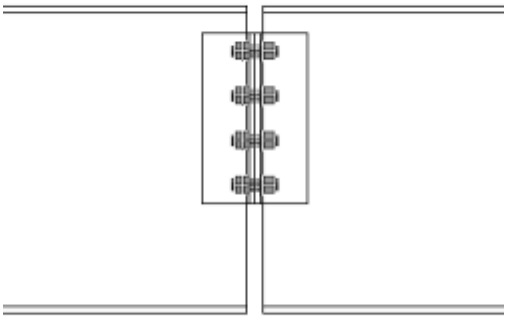
Girt to column

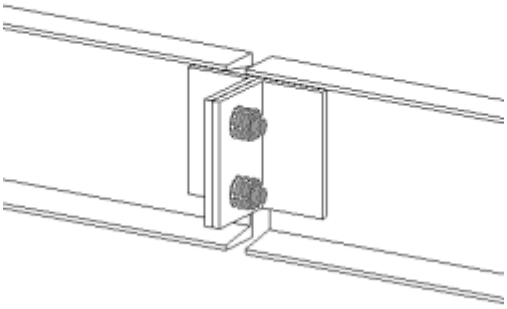
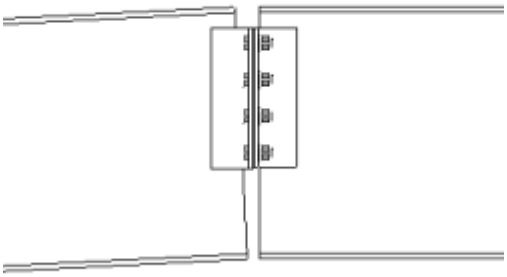
Example	Description
	Single girt to column. Angle connection. Use **Rail joint (70)**.
	Two girts to column. Angle connection. Use **Rail joint (70)**.
	Beam to column outrigger connection. Main and secondary part stiffener options. Use **Stub (28)**.

Example	Description
	Two girts to column. WT connection. Use **U.S. seat joint 3 (74)**.

Splice connections

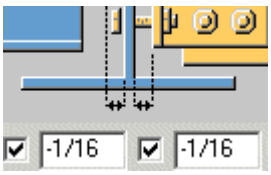
Beam to beam

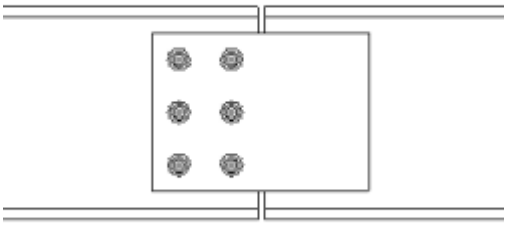
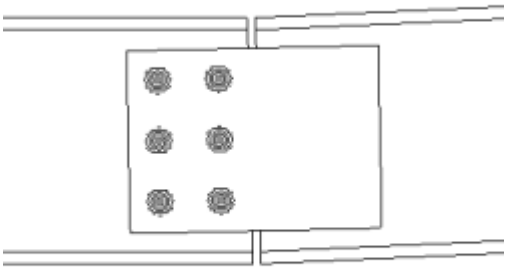
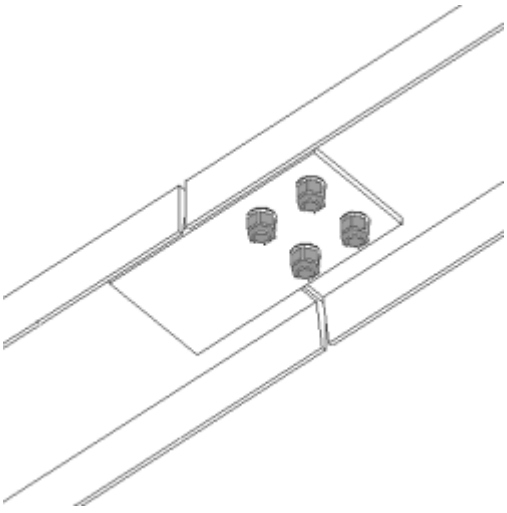
Example	Description
	Clip angle splice connection. Bolted/ bolted, welded/bolted, welded/ welded options. Use **Two sided clip angle (143)**.

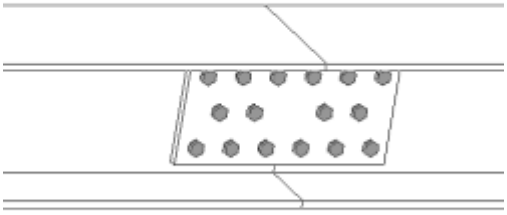
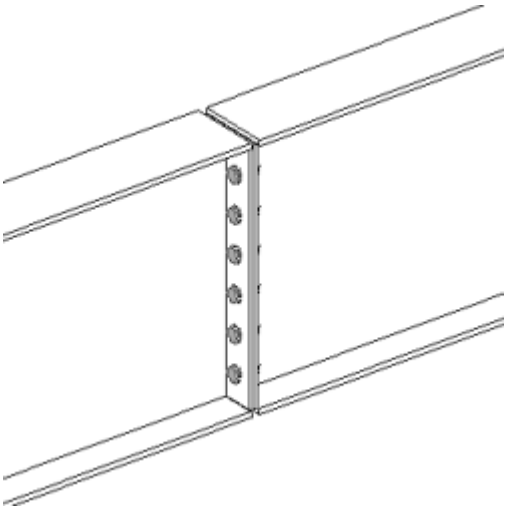
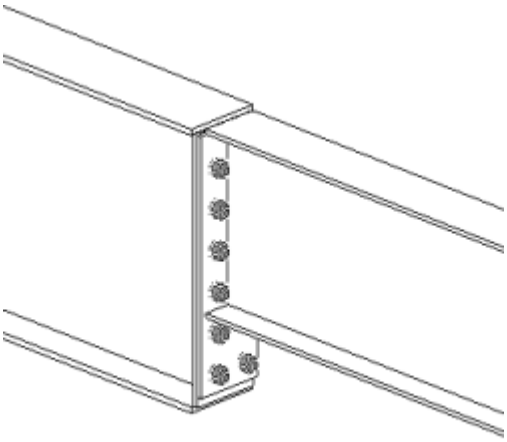
Example	Description
	Clip angle splice connection. One sided. Bolted/bolted, welded/bolted, welded/welded options. Use **Two sided clip angle (143)**.
	Clip angle splice connection. Beams in differing planes. Use **Two sided clip angle (143)**.

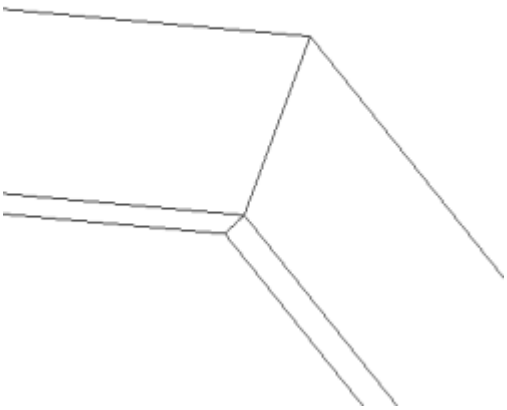
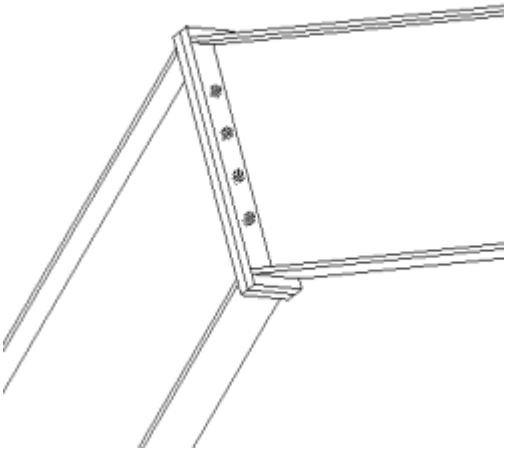
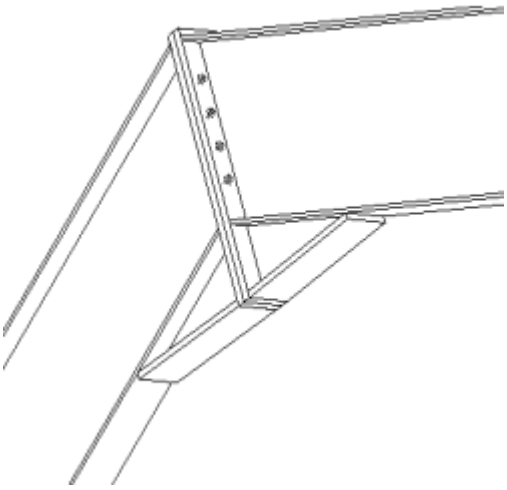
NOTE To make a clip angle type splice connection, it is necessary to create a dummy main part to bolt through. The secondary parts are spliced.

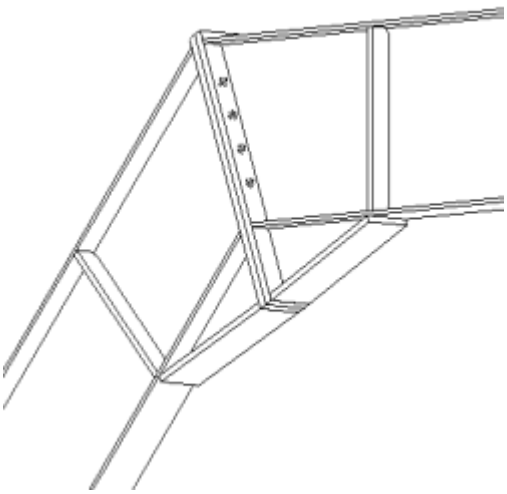
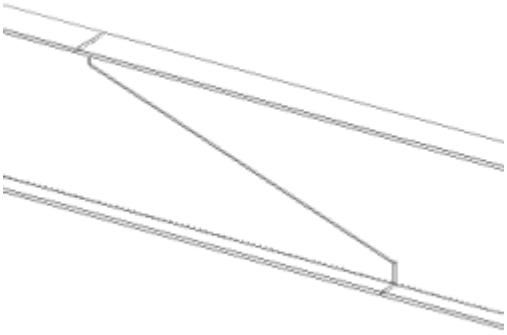
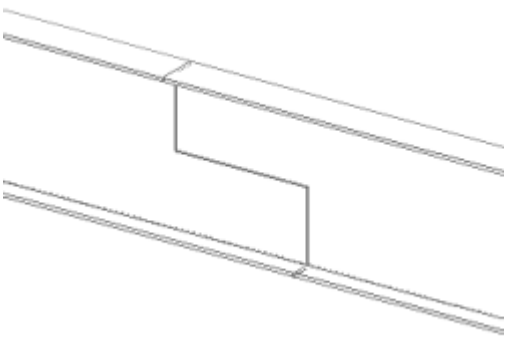
Make the dummy plate 1/8" thick with special properties so that it can be filtered out when making drawings. Use the settings shown below on the **Picture** tab.



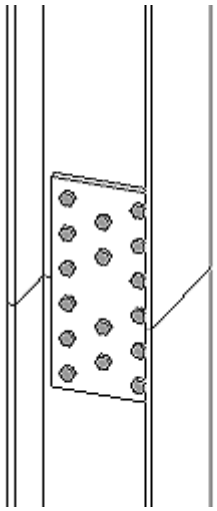
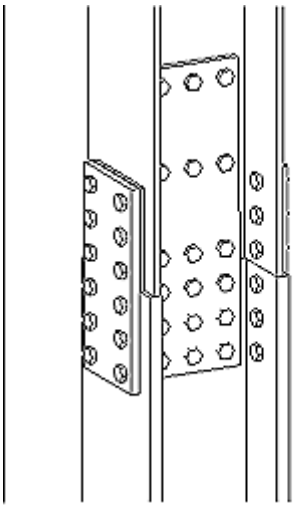
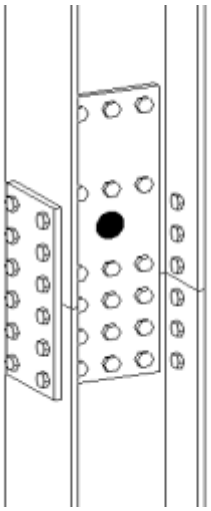
Example	Description
	Beam to beam web splice plate welded to main part/ bolted to secondary part. Use **Tab plate (33)**.
	Beam to beam web splice plate welded to main part/ bolted to secondary part. Beams in differing planes. Use **Tab plate (33)**.
	Beam to beam web splice plate welded to main part/ bolted to secondary part. Parts rotated. Use **Tab plate (33)**.

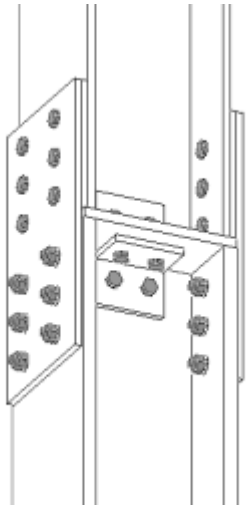
Example	Description
	Beam to beam web splice plate bolted to both parts. Both parts in the same plane. Use **Column splice (42)**.
	Beam to beam end plate splice. Use **Joining plates (14)**.
	Beam to beam end plate splice. Parts of different depths. Use **Joining plates (14)**.

Example	Description
	Beam to beam welded splice. Stair stringer to landing. Use **Cranked beam (41)**. **Note**: Does not work on very shallow slopes, or beams in the same plane.
	Beam to beam bolted end plate splice. Use **Cranked beam (41)**. **Note**: Does not work on very shallow slopes, or beams in the same plane.
	Beam to beam bolted end plate splice with haunch. Use **Cranked beam (41)**. **Note**: Does not work on very shallow slopes, or beams in the same plane.

Example	Description
	Beam to beam bolted end plate splice with haunch stiffener plate option. Use **Cranked beam (41)**. **Note**: Does not work on very shallow slopes, or beams in the same plane.
	Beam to beam welded splice, "Z" cut. Use **Offshore Z (192)**.
	Beam to beam welded splice, "Z" square cut. Use **Offshore Z (192)**.

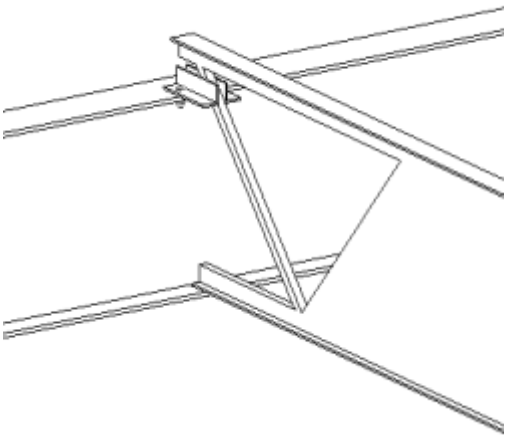
Column splice

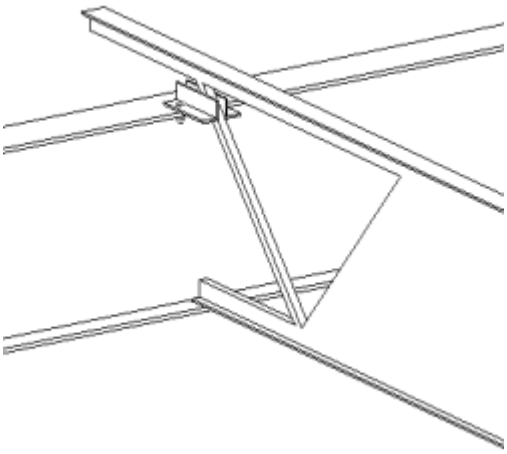
Example	Description
	Column splice. Bolted to both parts. Both parts in the same plane. Use **Column splice (42)**.
	Column splice. Bolted to both parts. Shim plates for differing profiles. Use **Column splice (42)**.
	Column splice. Bolted to both parts. Lifting hole option. Use **Column splice (42)**.

Example	Description
	Column splice with division plate and connection angles. Use **Column splice (132)**.

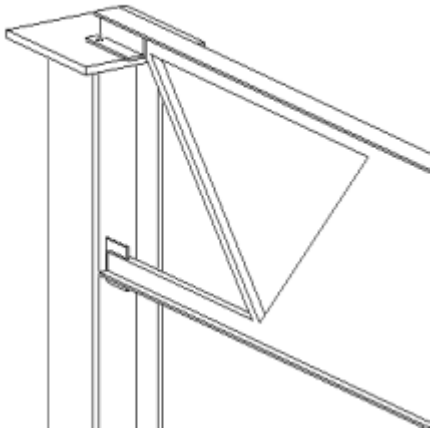
Joist connections

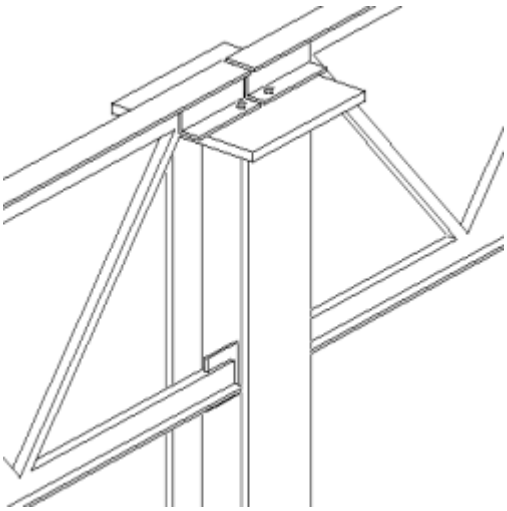
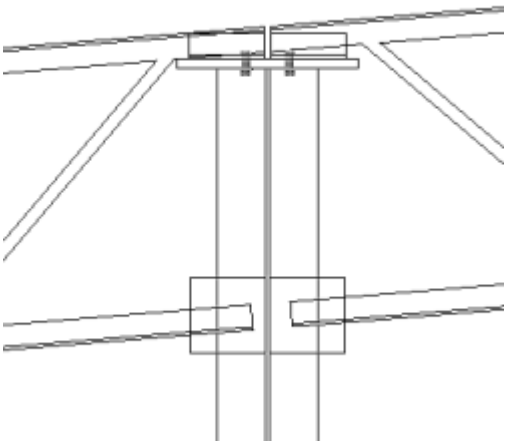
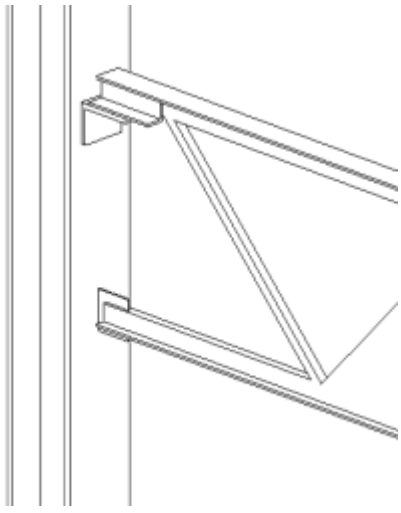
Joist to beam

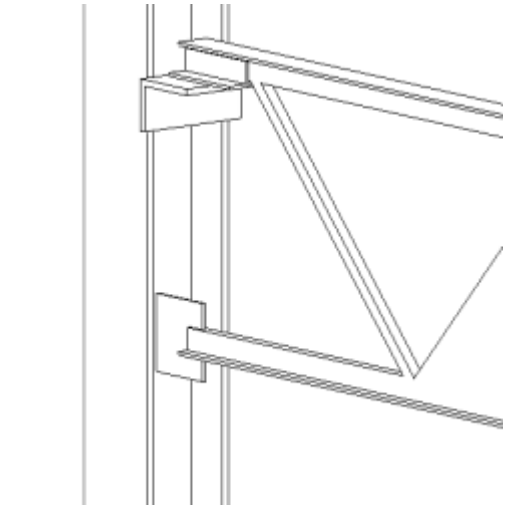
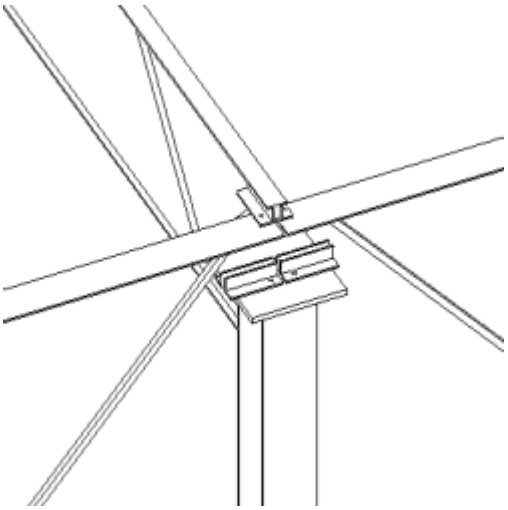
Example	Description
	Joist bearing on beam. Use **Joist to beam, type 1 (160)**.

Example	Description
	Joist bearing on beam. Top chord extension option. Use **Joist to beam, type 1 (160)**.

Joist to column

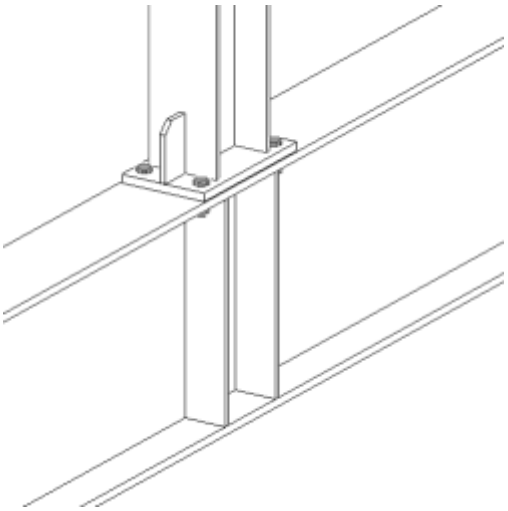
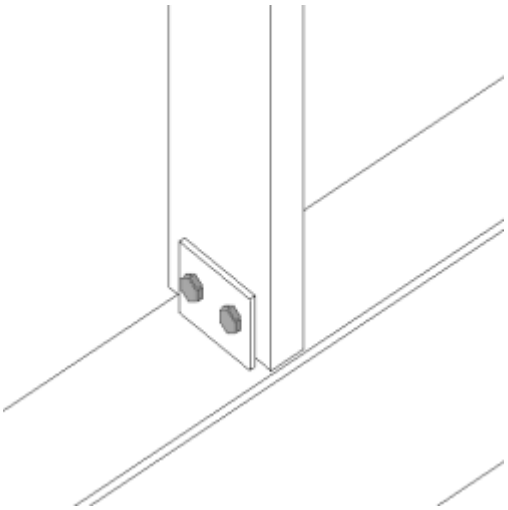
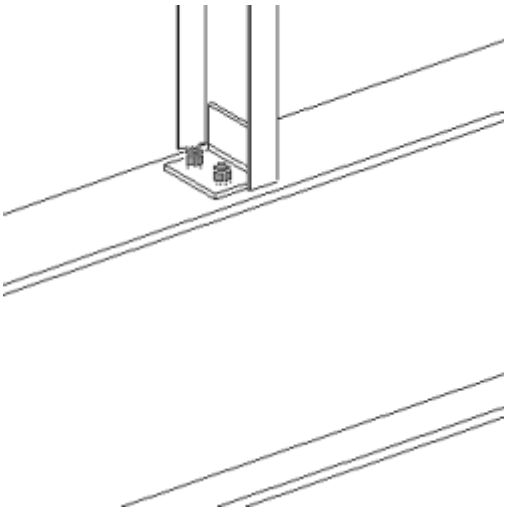
Example	Description
	Joist bearing to column. Cap plate, stabilizer bar or angle option. Top chord extension option also available. Use **Joist to column, type 1 (161)**.

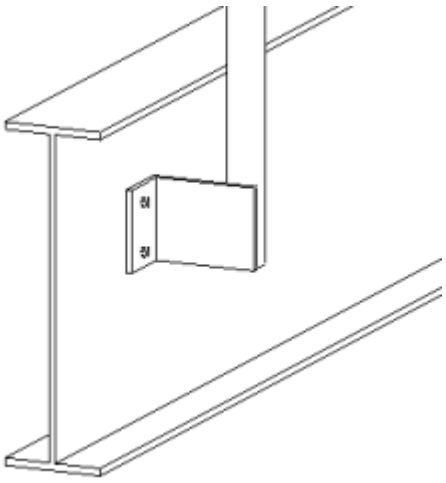
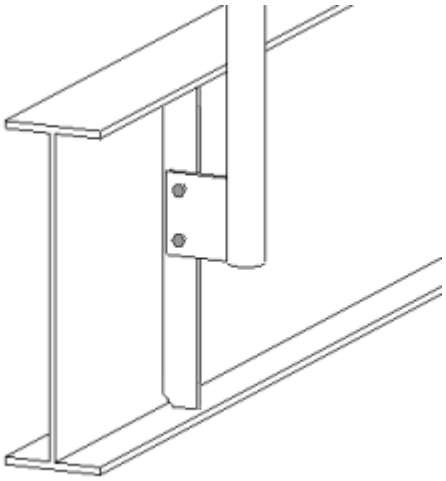
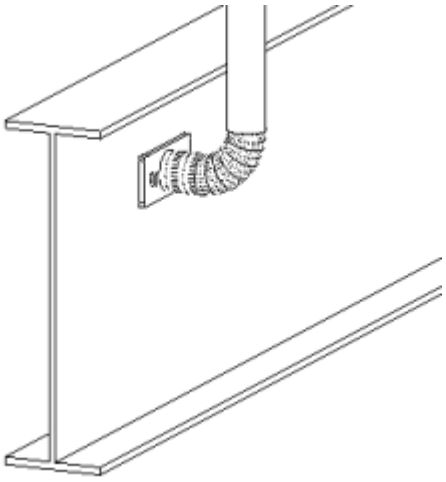
Example	Description
	Two joists bearing to column. Cap plate, stabilizer bar or angle option. Use **2 sided joist to column (162)**.
	Two joists bearing to column. Cap plate, cap plate level or on slope of joists. Use **2 sided joist to column (162)**.
	Joist framing to side of column. Stabilizer bar or angle option. Use **Joist to column, type 2 (163)**.

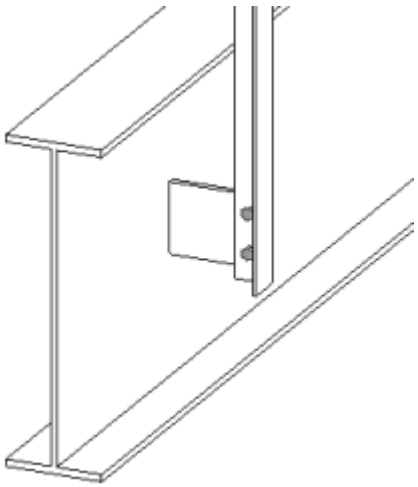
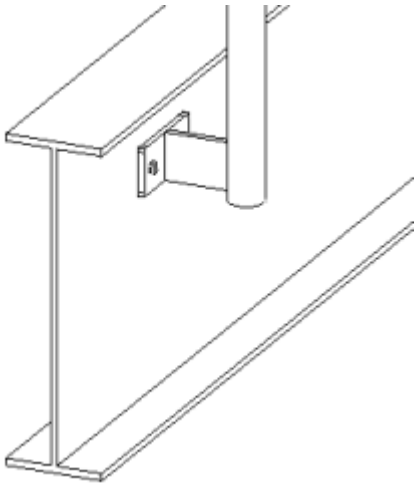
Example	Description
	Joist framing to side of column. Seat angle across toes of flanges. Use **Joist to column, type 2 (163)**.
	Joist framing to joist girders at column. Use **Joist to beam and column (164)**.

Vertical member to beam

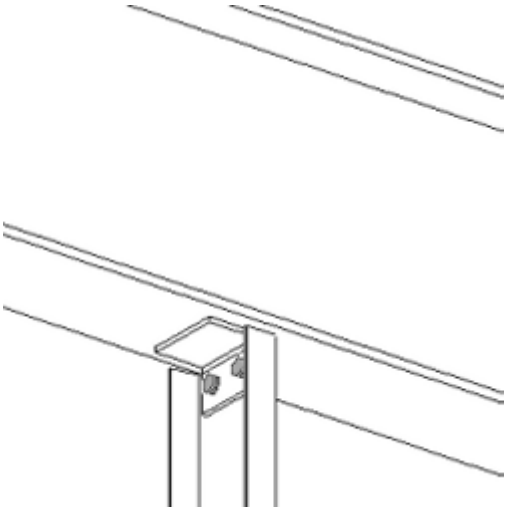
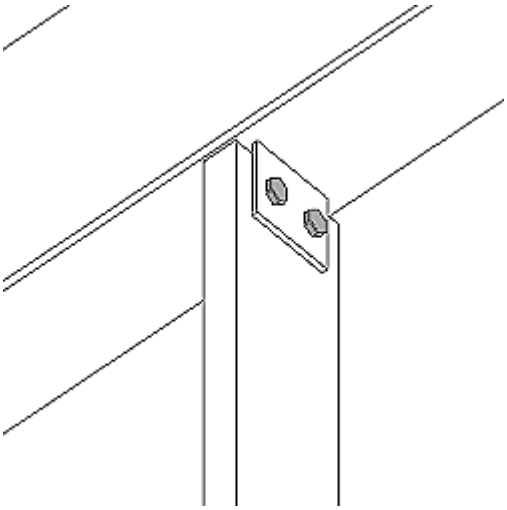
Post and door jamb to top of beam

Example	Description
	Post base plate to top of beam. Main and secondary stiffener options. Use **U.S. base plate connection (71)**.
	Simple shear tab to post or channel jamb. Use **Shear plate simple (146)**.
	Clip angle to channel jamb. Use **Clip angle (141)**.

Example	Description
	Clip angle to railing post. Use **Stringer stanchion L profile (68)**.
	Railing post connection plate to beam stiffener. Use **Stringer stanchion st (69)**.
	Railing post elbow to connection plate. Use **Stanchion curved (84)**.

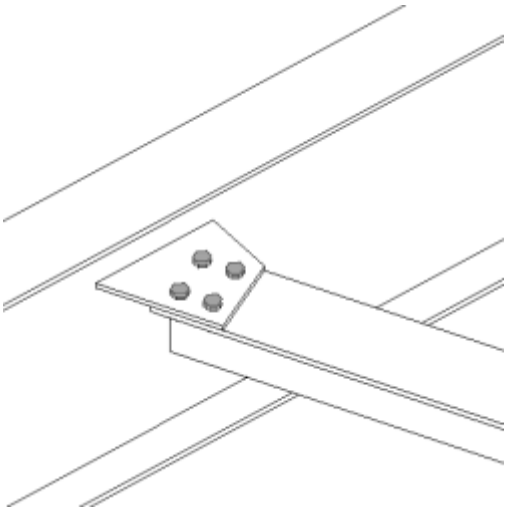
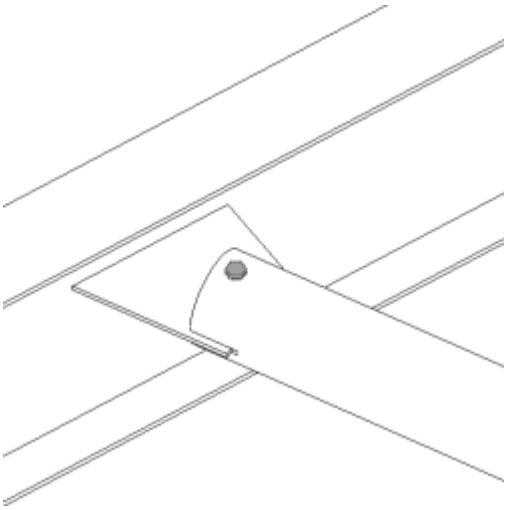
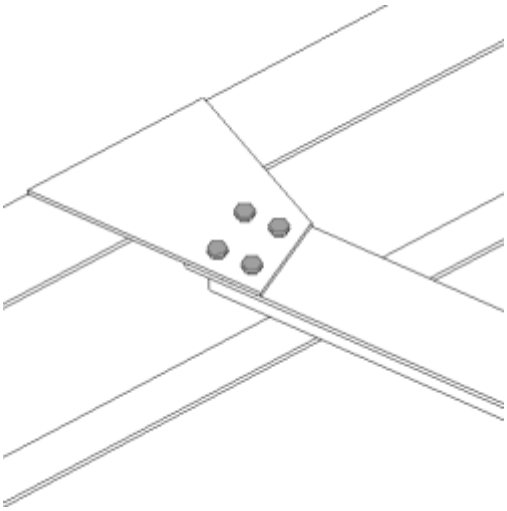
Example	Description
	Shear tab to railing post. Use **Stanchion side profile (86)**.
	Built–up plate railing post connection. Use **Stanchion double plate (87)**.

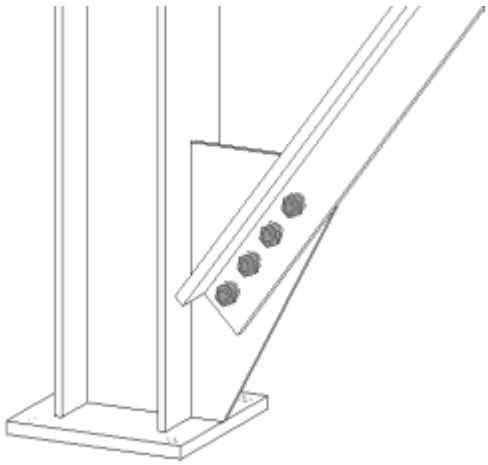
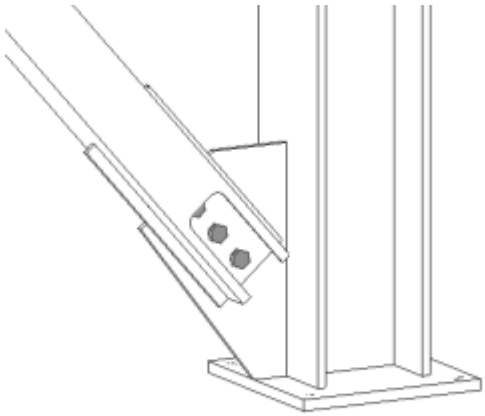
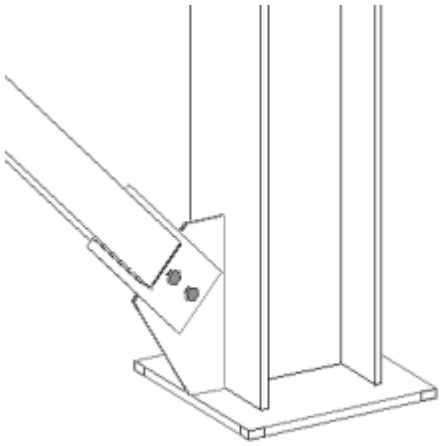
Hangers from underside of beam

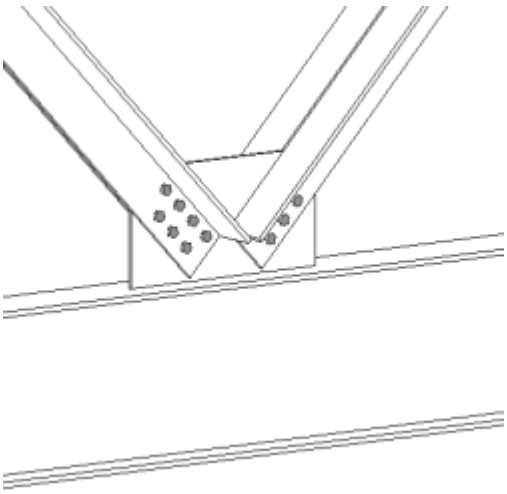
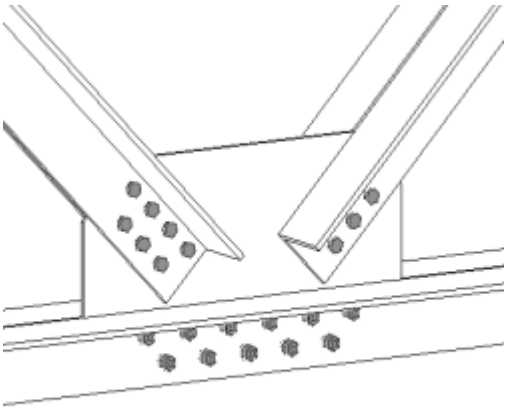
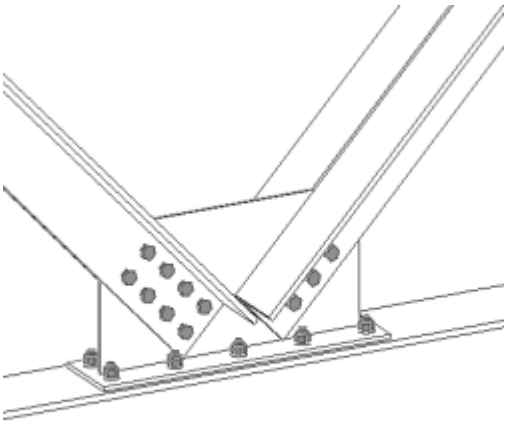
Example	Description
	Clip angle hanger connection. Use **Clip angle (141)**.
	Simple shear tab hanger connection. Use **Shear plate simple (146)**.

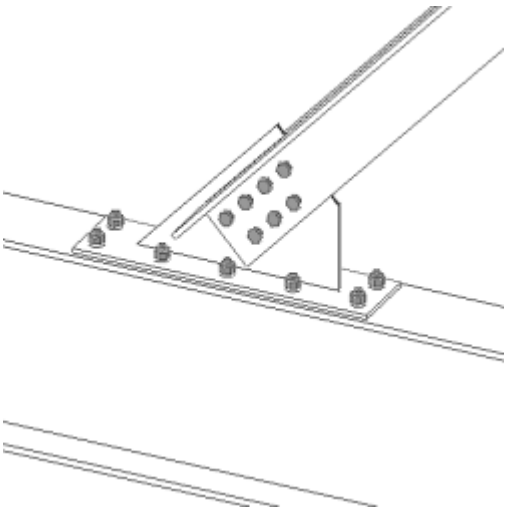
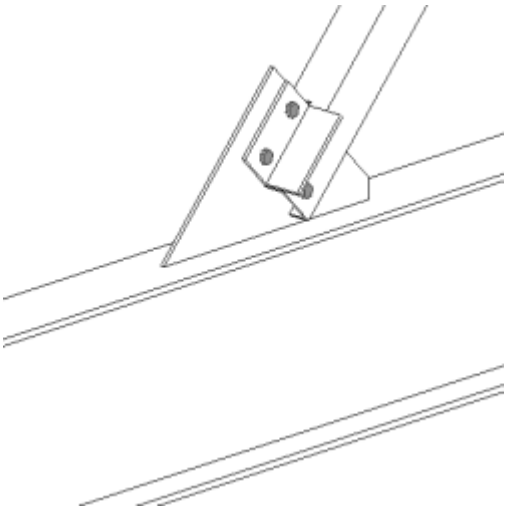
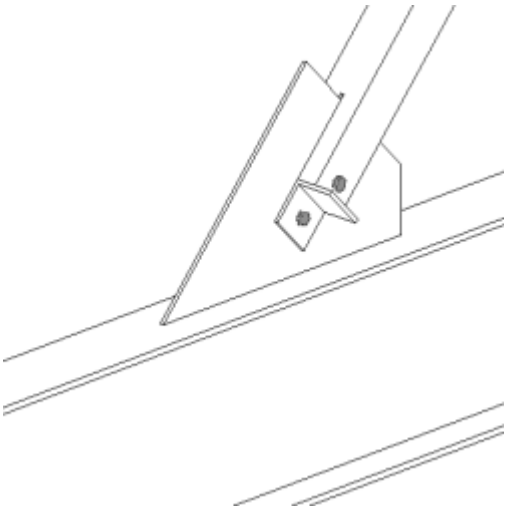
Bracing connections

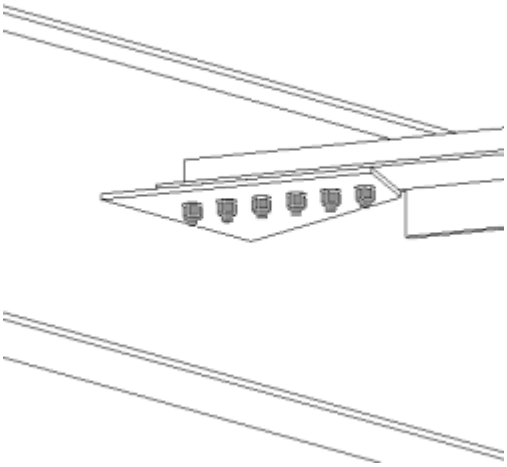
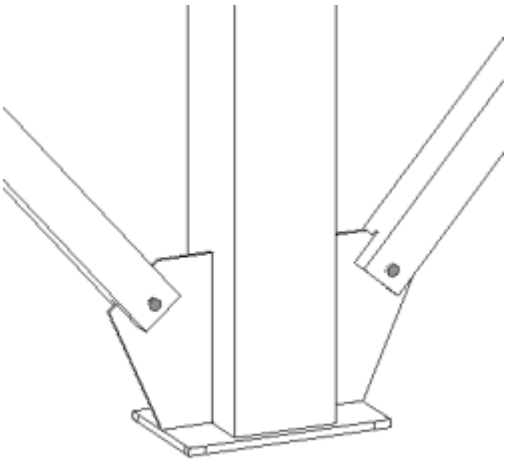
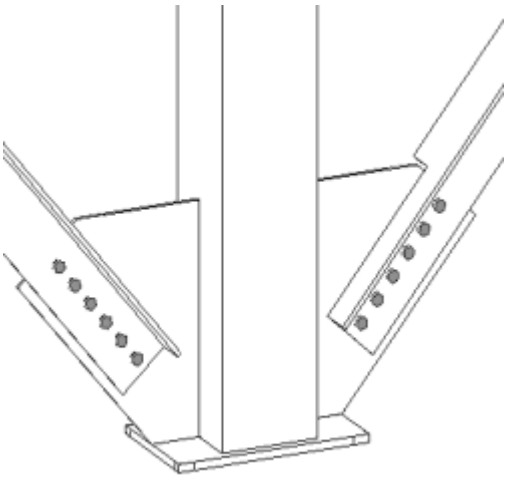
Simple gusset plate connections

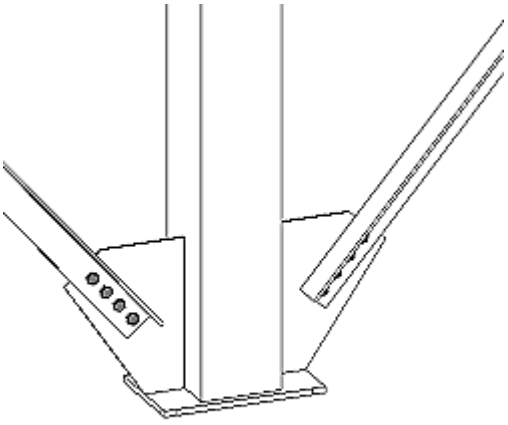
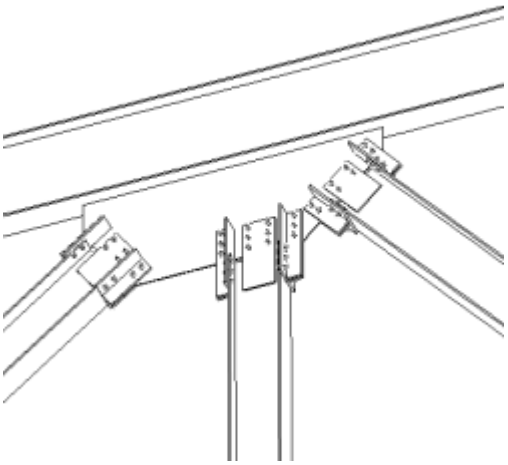
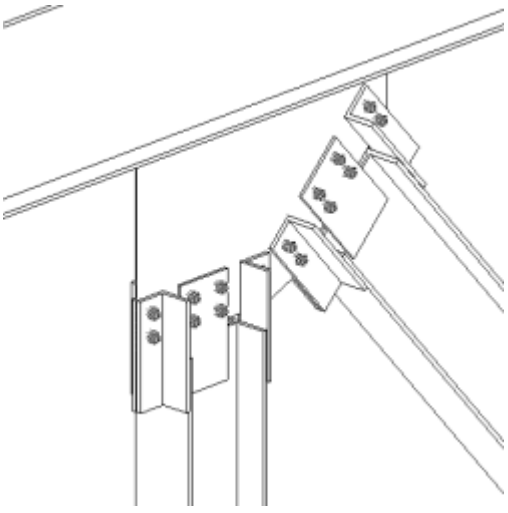
Example	Description
	Gusset plate to single brace. Horizontal and vertical bracing. Various bracing profiles. Use **Bolted gusset (11)**. Load connection attribute **< Defaults >** and select **Defaults** for **Rule group** for best results.
	Gusset plate to single brace. Horizontal and vertical bracing. Hollow round brace with pin bolt option. Use **Bolted gusset (11)**. Load connection attribute **< Defaults >** and select **Defaults** for **Rule group** for best results.
	Gusset plate to single brace. Brace and main part at the same elevation. Use **Bolted gusset (11)**.

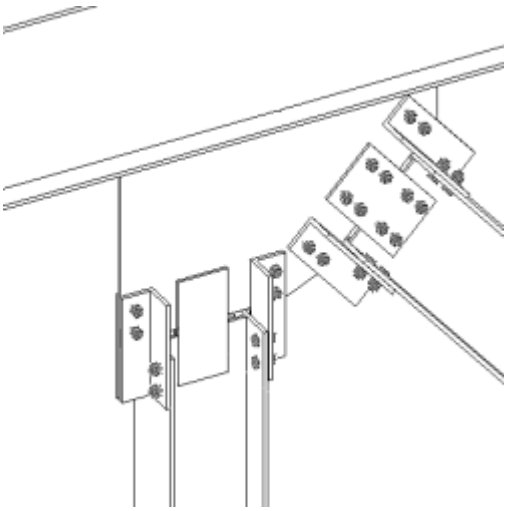
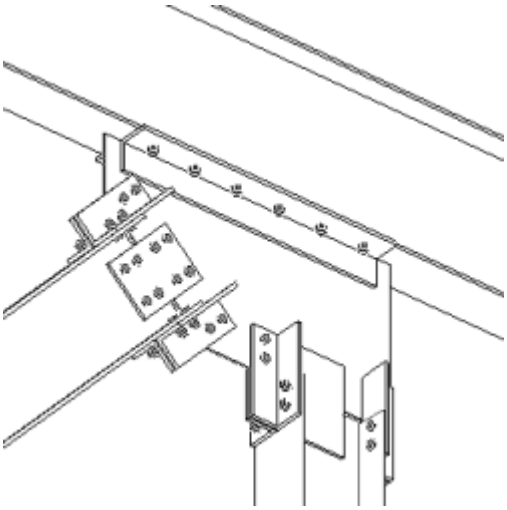
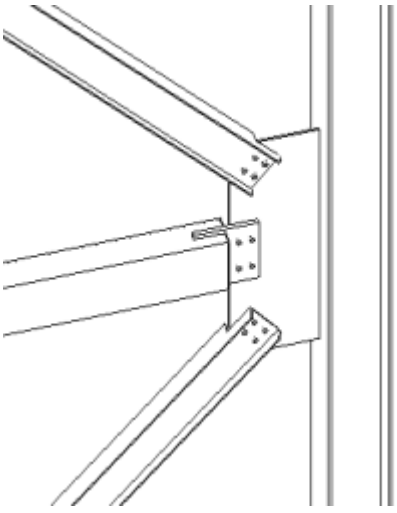
Example	Description
	Gusset plate to single brace at base plate of column. Use **Bolted gusset (11)**.
	Gusset plate hollow section "Birds mouth" connection to secondary parts. Use **Tube gusset (20)**.
	Gusset plate hollow section tongue plate connection to secondary parts. Use **Tube gusset (20)**.

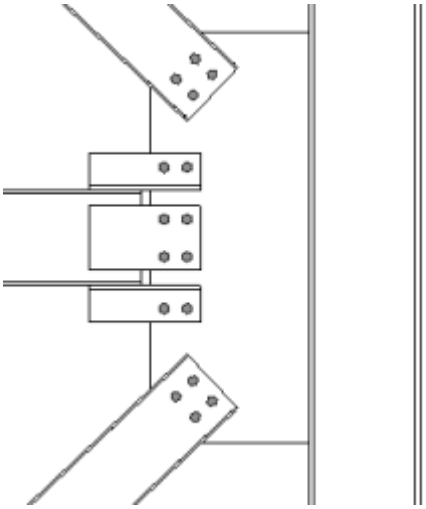
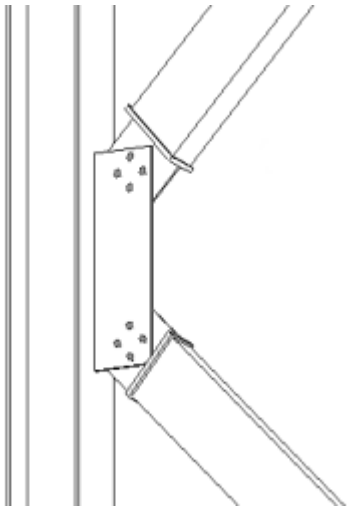
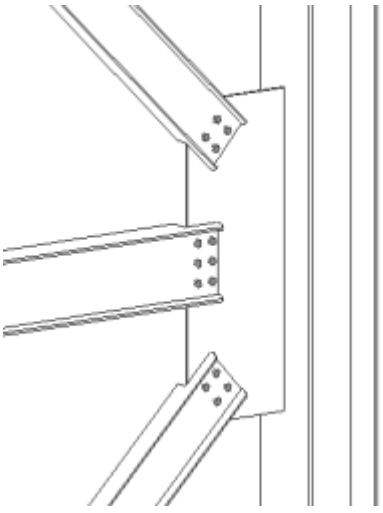
Example	Description
	Gusset plate to twin profile brace. Vertical or horizontal brace. Multiple bracing members. Use **Bolted gusset (11)**.
	Gusset plate to twin profile brace. Twin profile main part. Welded or bolted to main part. Use **Bolted gusset (11)**.
	Gusset plate with connection plate. Bolted to main part. Use **Bolted gusset (11)**.

Example	Description
	Gusset plate with connection plate. Bolted to main part. Various gusset plate shaping options. Use **Bolted gusset (11)**.
	Gusset plate to hollow section brace. Pin bolt and tension angle option. Use **Bolted gusset (11)**.
	Gusset plate to hollow section brace. Pin bolt and tension angle option. Use **Bolted gusset (11)**.

Example	Description
	Gusset plate to WT section brace. Notching option. Use **Bolted gusset (11)**. Load connection attribute **< Defaults >** and select **Defaults** for **Rule group** for best results.
	Gusset plate through hollow section column to hollow section bracing at base plate. Use **Bolted gusset (11)**. Select column, then brace and brace.
	Gusset plate through hollow section column to WT bracing at base plate. Use **Bolted gusset (11)**. Select column, then brace and brace.

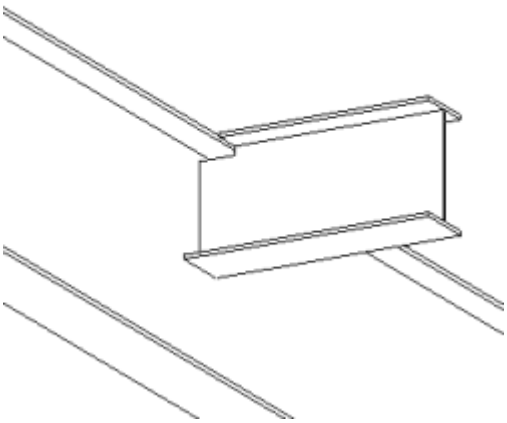
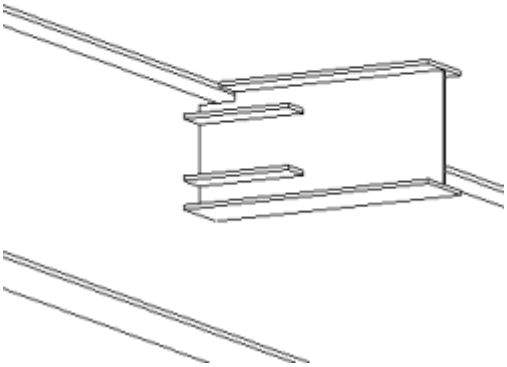
Example	Description
	Gusset plate through hollow section column to angle bracing at base plate. Single or twin profile. Use **Bolted gusset (11)**. Select column, then brace and brace.
	Wrapped gusset plate W section bracing. Various bracing connection options. Welded/bolted, bolted/ bolted. Use **Gusset wrapped cross (62)**.
	Wrapped gusset plate W section bracing. Various bracing connection options. Welded/bolted, bolted/ bolted. Use **Gusset wrapped cross (62)**.

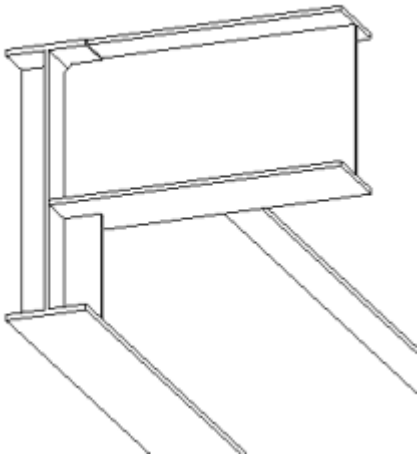
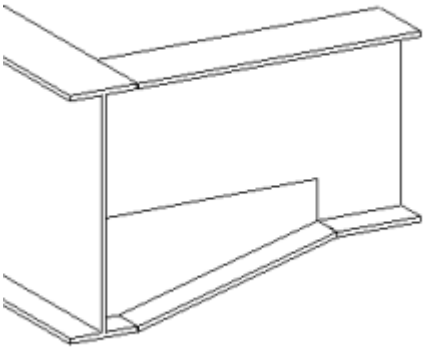
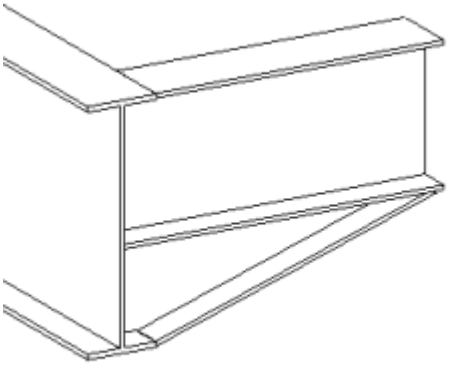
Example	Description
	Wrapped gusset plate W section bracing. Various bracing connection options for each brace. Use **Gusset wrapped cross (62)**.
	Wrapped gusset plate W section bracing. Various gusset plate connection options. Use **Gusset wrapped cross (62)**.
	Hollow section bracing tongue plate connection to existing gusset plate. Use **Tube crossing (22)**.

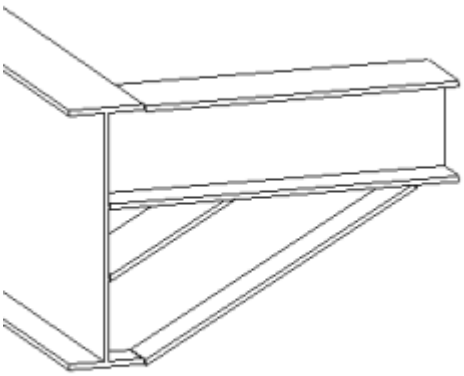
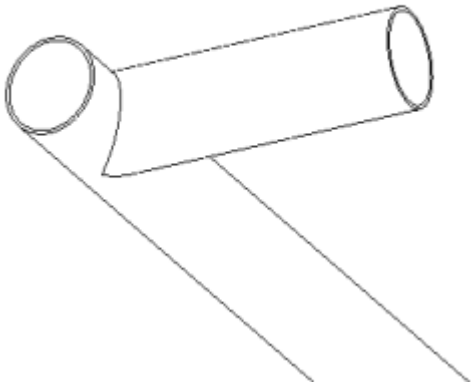
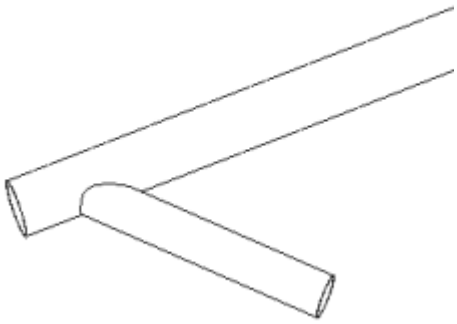
Example	Description
	Wrapped gusset W section bracing connection to existing gusset plate. Use **Wrapped cross (61)**.
	Hollow section bracing WT end connection to existing gusset plate. Use **Portal bracing (105)**.
	Bolted brace connection to existing gusset plate. Use **Bracing cross (19)**.

Welded connections

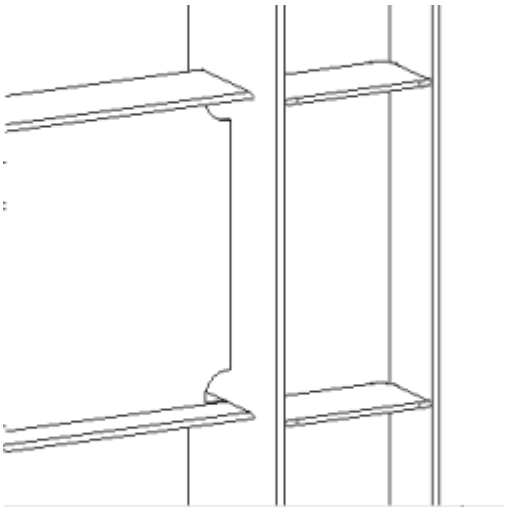
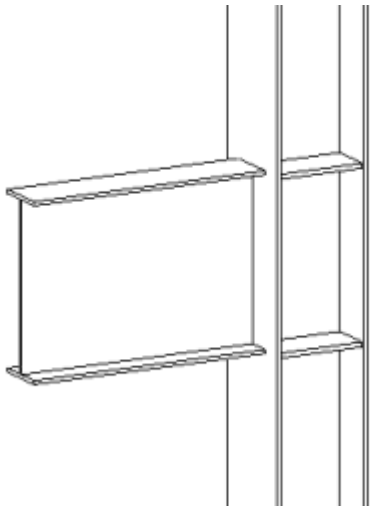
Beam to beam

Example	Description
	Welded beam to beam. Use **Fitting (13)**.
	Welded beam to beam with horizontal stiffeners. Use **Fitting (13)**.

Example	Description
	Welded beam to beam with primary. Vertical stiffeners. Use **Welded beam to beam (123)**.
	Welded beam to beam with haunch. Use **Offshore (194)**.
	Welded beam to beam with haunch. Various haunch options. Use **Offshore (194)**.

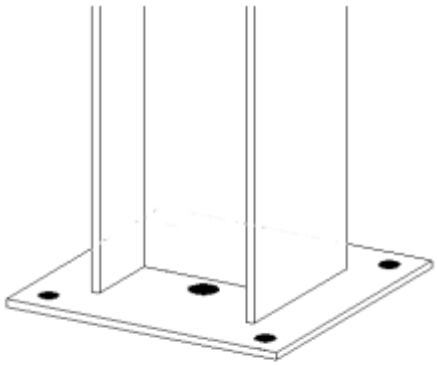
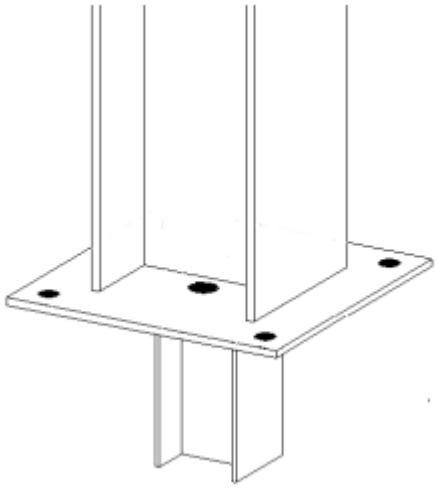
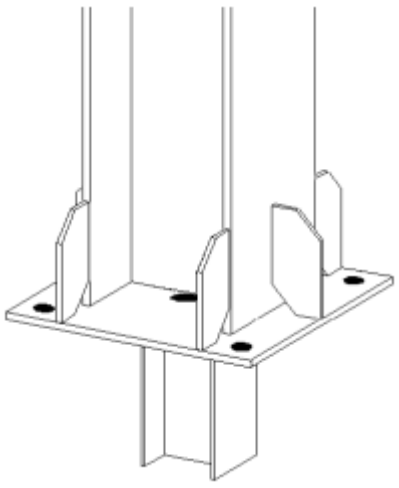
Example	Description
	Welded beam to beam with haunch. Various haunch options. Use **Offshore (194)**.
	Welded round profile to round profile. Use **Round tube (23)**.
	Welded round profile to round profile. Differing profile size and skewed secondary part. Use **Round tube (23)**.

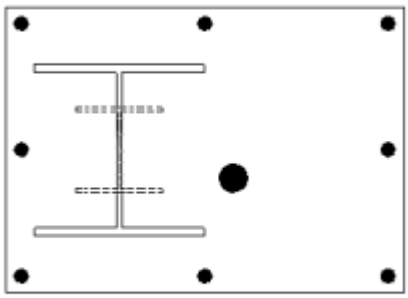
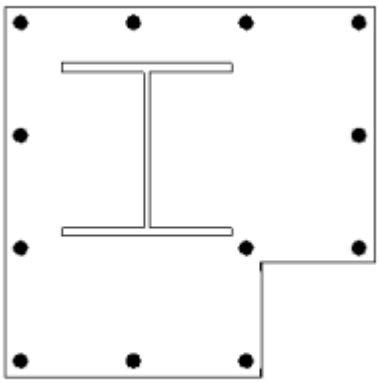
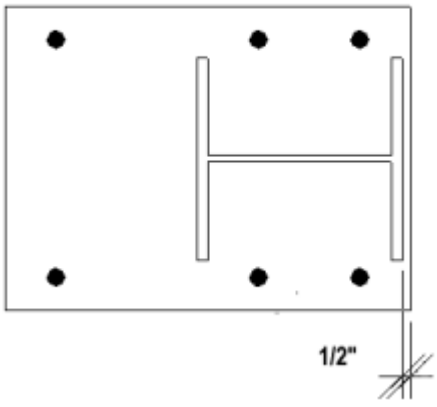
Beam column

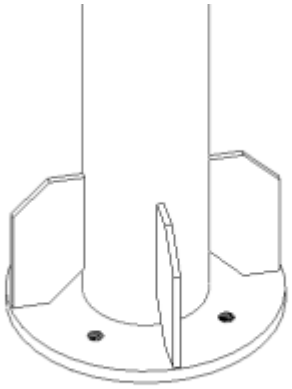
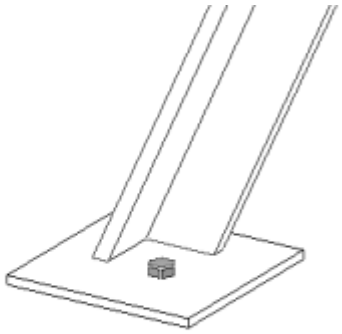
Example	Description
	Welded to column with weld preparation and stiffener options. Use **Welded column with stiffeners (128)**.
	Welded to column. Use **Welded column (31)**.

Details

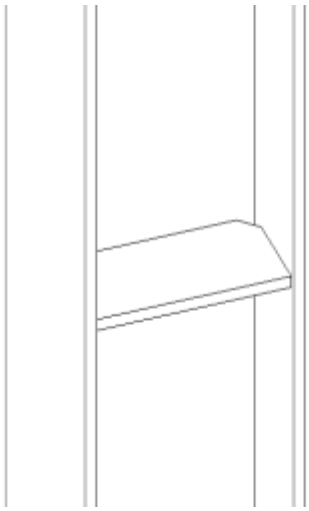
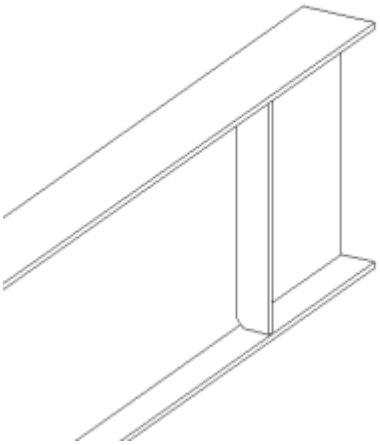
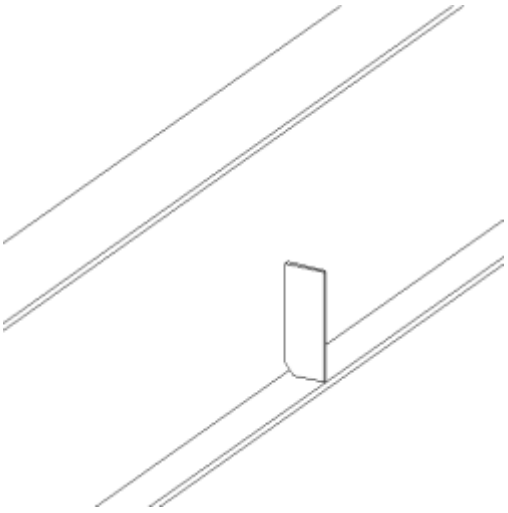
Base plates

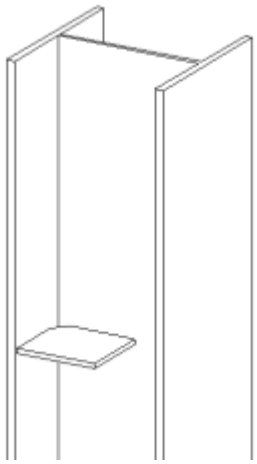
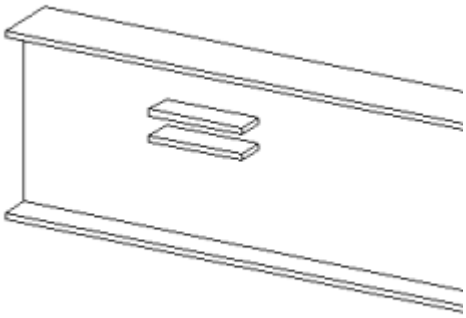
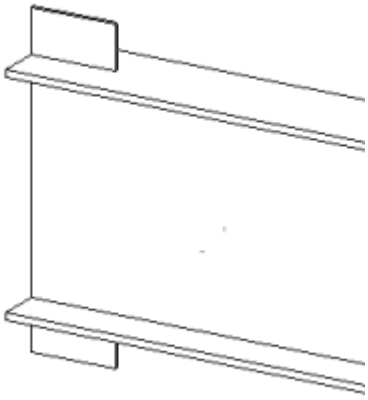
Example	Description
	Column base plate with grout hole option. Use **U.S. base plate (1047)**.
	Column base plate with shear key option. Use **U.S. base plate (1047)**.
	Column base plate with stiffener option. Use **U.S. base plate (1047)**.

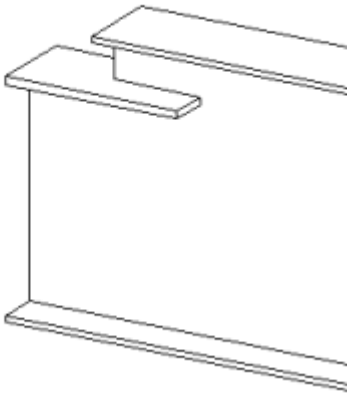
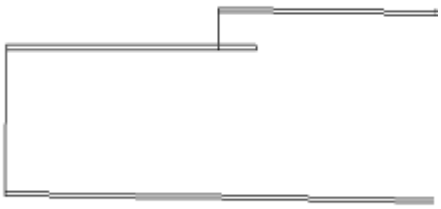
Example	Description
	Offset column base plate (shear key and grout hole optional). Use **U.S. base plate (1047)**.
	Column base plate bolt elimination and interior corner chamfer option. Use **U.S. base plate (1047)**.
	Column base plate located from face of flange. Use **Base plate (1042)**.

Example	Description
	Circular column base plate with stiffener option. Use **Circular base plates (1052)**.
	Horizontal base plate to sloped post. Use **Base plate (1053)**.

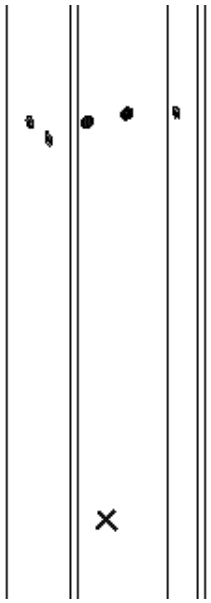
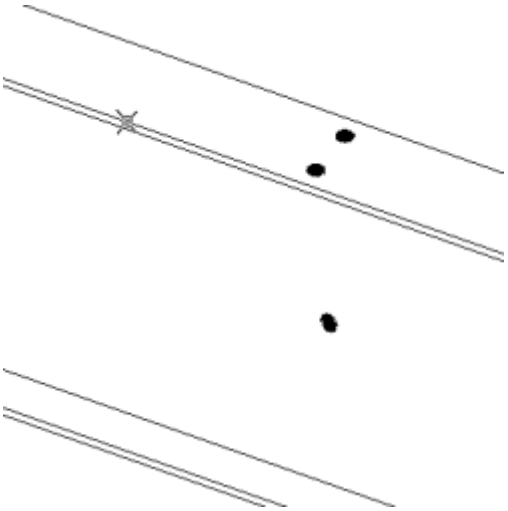
Stiffeners

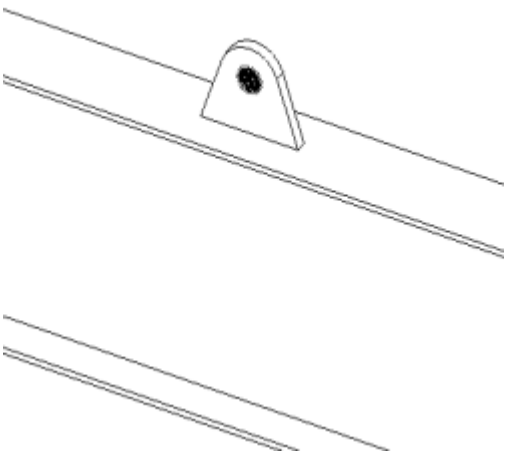
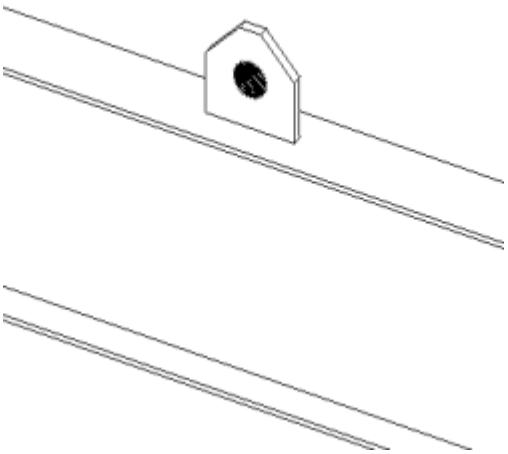
Example	Description
	Stiffener plate detail to column. Use **Stiffeners (1003)**.
	Stiffener plate detail to beam. Use **Stiffeners (1003)**.
	Partial depth stiffener plate detail to beam. Use **Stiffeners (1041)**.

Example	Description
	Partial depth stiffener plate detail to column. Use **Stiffeners (1041)**.
	Parallel stiffener plates. Use **Horizontal stiffener (1017)**.
	Flange stiffener plates. Use **Stiffeners (1030)**.

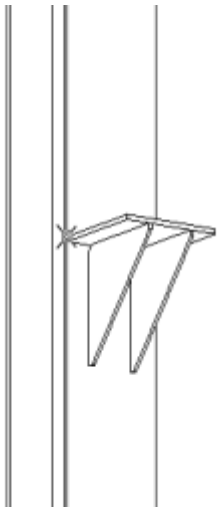
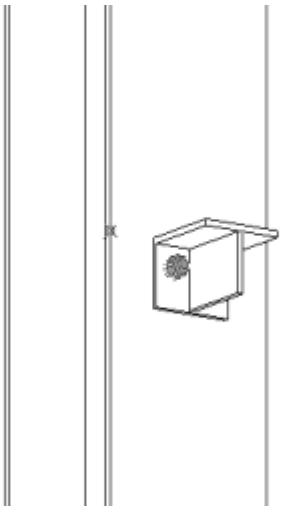
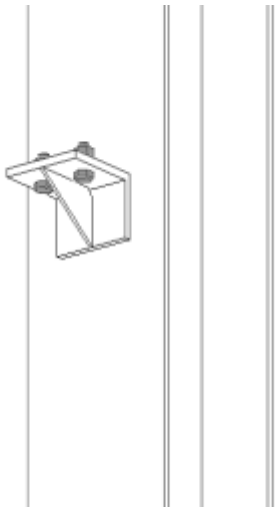
Example	Description
	Stiffened notch. Use **Stiffened notch (1006)**.
	Stiffened notch. Notch cut bevel to part. Use **Stiffened notch (1006)**.

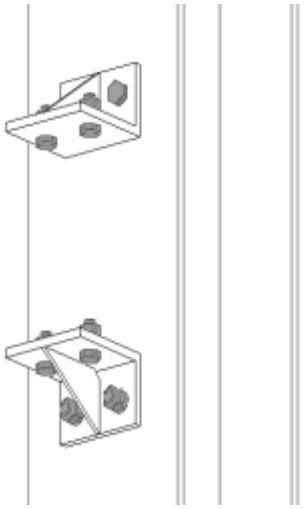
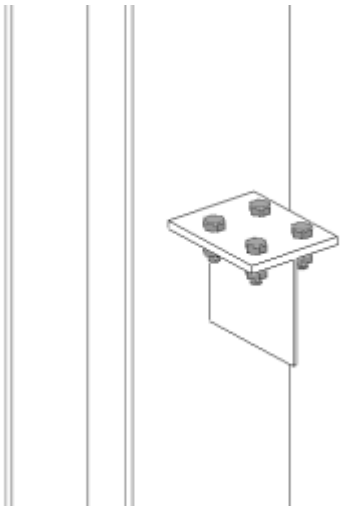
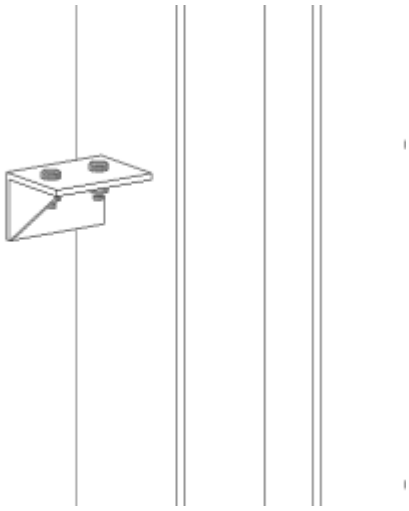
Manlock holes and lifting lugs

Example	
	Manlock holes in column. Use **Manlock column (1032)**.
	Manlock holes in beam. Use **Manlock beam (1033)**.

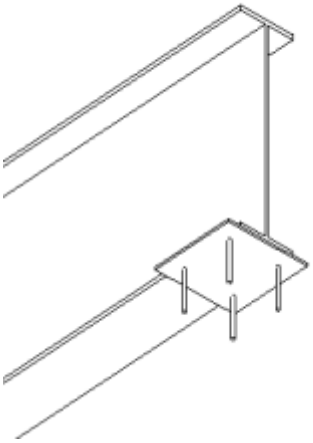
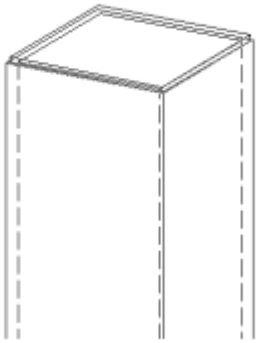
Example	
	Lifting lug to beam. Use **Lifting/alignment pieces (1031)**.
	Lifting lug to beam chamfered plate. Use **Lifting/alignment pieces (1031)**.

Seat details

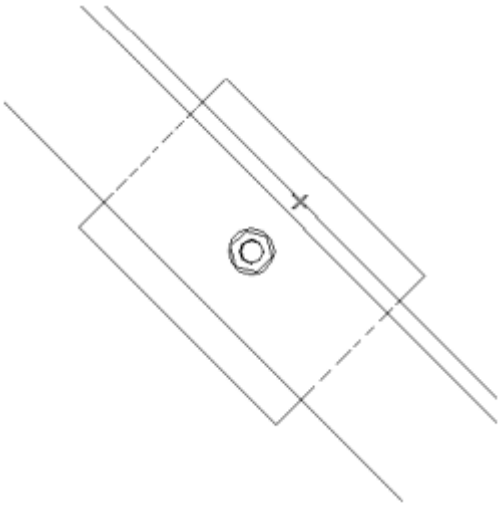
Example	Description
	Plate seat with stiffeners. Use **Stub plate (1013)**.
	Angle seat with stiffener. Use **Angle profile box (1040)**.
	Angle seat with stiffener. Bolt to main part and through seat options. Use **U.S. seat detail (1048)**.

Example	Description
	Angle seat with stiffener. Bolt to main part placed at a given distance apart. Use **U.S. seat detail (1048)**.
	WT seat detail. Use **U.S. seat detail 2 (1049)**.
	Rotated angle seat. Stiffener plate option available. Use **U.S. seat detail 2 (1049)**.

Cap plate and bearing plate

Example	Description
	Bearing plate at end of beam. Use **U.S. bearing plate (1044)**.
	Cap plate. Use **End plate detail (1002)**.

Miscellaneous

Example	Description
	Spacer plate between twin profiles. Welded or bolted. Use **Twin profile connection plate (1046)**.

6 Concrete components reference

This section contains information on the use of concrete components that ship with Tekla Structures.

If you know which component you need, you can press F1 in the component dialog box to quickly access the correct help page. Some components use locally installed help files in a legacy format, which you can only access by pressing F1 in the component dialog box.

More components are available in Tekla Warehouse for you to download and install.

You can also modify many of the existing components and create your own custom components, see .

6.1 Concrete detailing

This section introduces the concrete detailing tools available in Tekla Structures.

Click the links below to find out more:

- Seating connections (page 3321)
- Beam and column connections (page 3348)
- Panels and walls (page 3438)
- Formwork placing tools (page 3587)
- Openings (page 3696)
- Flooring (page 3699)
- Concrete stairs (page 3759)
- Foundations (page 3811)

NOTE Precast components can only be applied to precast cast units. They cannot be applied to cast-in-place cast units.

Seating connections

Tekla Structures includes several seating connections that you can use to connect concrete columns and beams using anchor bolts. The seating connection tools are:

- Seating with dowel (75) (page 3321)
- Two-sided seating with dowel (76) (page 3328)
- Seating with dowel to flange (77) (page 3336)
- Two-sided seating with dowel to flange (78) (page 3342)

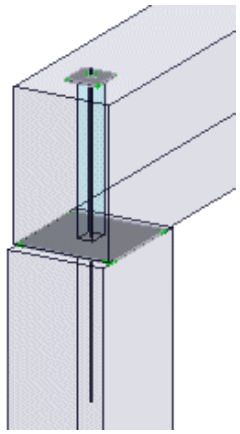
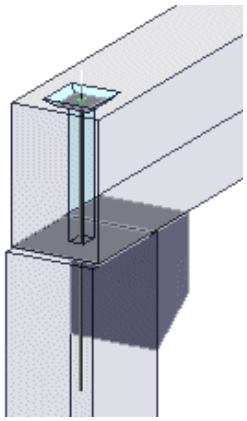
Seating with dowel (75)

Seating with dowel (75) connects a column and a beam using an anchor bolt.

Parts created

- Anchor bolt
- Nut
- Washer plate
- Bearing plate
- Fittings for beam and column
- Hole for bolt
- Corbel (optional)
- Recess for nut and washer plate (optional)

Use for

Situation	Description
	Connects a beam and a column using an anchor bolt. Washer plate and nut protrude from the beam.
	Connects a beam and a column using an anchor bolt and a beveled corbel. Washer plate and nut recessed into the beam.

Before you start

Create the following parts:

- Concrete column (round or rectangular profile)
- Concrete beam (rectangular, HI, I, L, or inverted T profile)

Selection order

1. Select the main part (column).
2. Select the secondary part (beam).

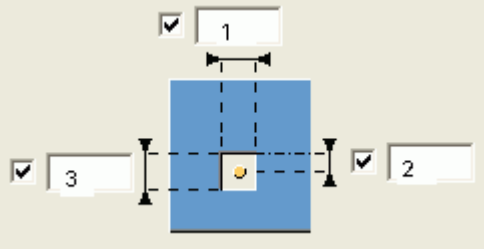
   The connection is created automatically when the second part is selected.

Picture tab

Use the **Picture** tab to define position of anchor bolt and bearing plate, bolt hole dimensions, grout type, and beam clearance.

Bolt hole

Enter the following bolt hole dimensions:



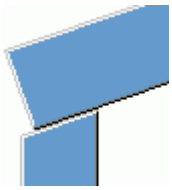
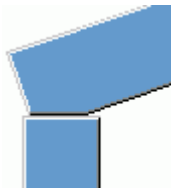
	Description
1	Hole dimension in the direction of the beam.
2	The distance from beam center line to hole center and bolt.
3	Hole dimension in the direction perpendicular to the beam.

Select the shape of the bolt hole:

Option	Description
	Rectangular Default
	Round

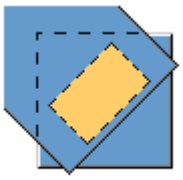
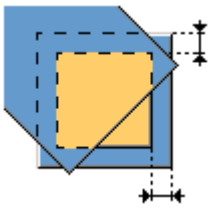
Beam and column

Select one of the following options to have Tekla Structures fit the column or the beam:

Option	Description
	Fits the column. Default
	Fits the beam. The beam must be sloped.

Bearing plate

Select one of the following options to define the position of the bearing plate:

Option	Description
	Square with beam. Default
	Square with column.
	Square with column. Enter the distances from column edges.

Dowel tab

Use the **Dowel** tab to define dowel and grout properties.

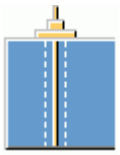
Dowel

Option	Description
Profile	Select the dowel profile from the profile catalog.
Prefix, **Start number**	Prefix and start number for the part position number.
Material	Material grade. The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options** .
Name	Name that is shown in drawings and reports.
Finish	Describes how the part surface has been treated.
Class	Enter a number to group the parts that the component creates. By

Option	Description
	default, the class number affects the color in which the part is shown in model views.
Size	Diameter of the bars.
Grade	Strength of the steel used in the bars.
Number of bars	Select **1 Dowel** to create one reinforcing bar. Select **2 Dowels** to create two reinforcing bars. Then define the distance between the bars in the **Bar distance** field.

Grout

Select one of the following options to include and define grout:

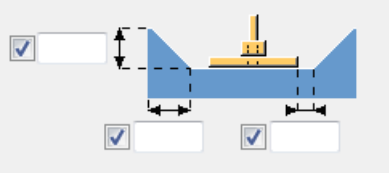
Option	Description
	No grout. Default
	Bolt hole grouted. No nut or washer plate.
	Bolt hole grouted. Bolt, washer plate, and anchor bolt protruding.

Nut and washer plate

Select one of the following options to define if the nut and washer plate are recessed into the beam:

Option	Description
	Nut and washer plate on the surface of the beam. Default
	Nut and bolt plate recessed into the beam.

If you choose to recess the nut and washer plate into the beam, enter the following dimensions to define the recess:

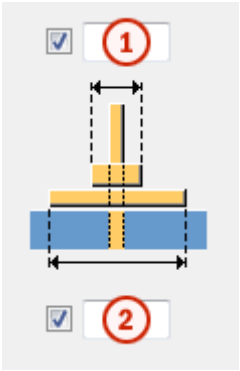


Parts tab

Use the **Parts** tab to define bearing pad, drainage hole, grout, bolt plate, nut, and tube properties.

Option	Description
t, **b**, **h**	Define the part thickness, width and height.
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name for the part.
Class	Use **Class** to group the parts.
Cast unit	Select to add the parts to the cast unit.

Nut and bolt plate



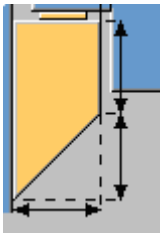
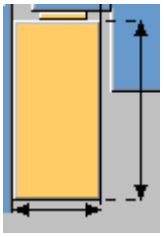
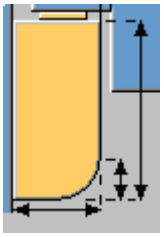
Field	Description
1	Nut width.
2	Bolt plate width.

Corbel tab

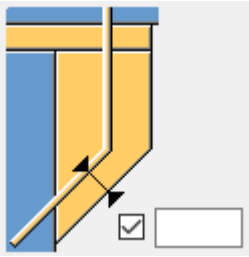
Use the **Corbel** tab to create a corbel and define its properties.

Select to create a corbel from the **Create corbel** list.

The options for chamfering corbels are:

Option	Description
	Beveled Default
	Straight
	Rounded

Define the placement of the corbel rebar.



Option	Description
t, b, h	Define the thickness, width, and height of the part.
Pos_No	Prefix and start number for the part position number. The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options** .

Option	Description
Name	Name that is shown in drawings and reports.
Class	Part class number.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

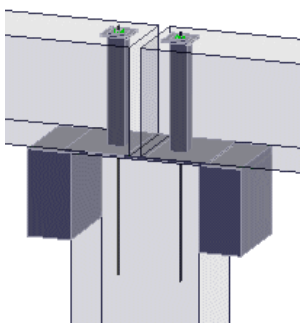
Two-sided seating with dowel (76)

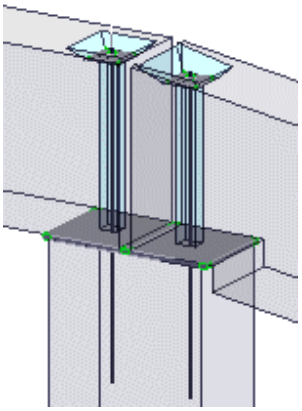
Two-sided seating with dowel (76) connects a column and two beams using anchor bolts.

Parts created

- Anchor bolts (2)
- Nuts (2)
- Washer plates (2)
- Bearing plates (2)
- Fittings for beam and column (2)
- Holes for bolts (2)
- Corbels (optional) (2)
- Recesses for washer plate and nut (optional) (2)

Use for

Option	Description
	Connects two beams to a column using anchor bolts and creates corbels. Nuts and washer plates on the surface of the beams.

Option	Description
	Connects two beams to a column using anchor bolts. Nuts and washer plates recessed into the beams. Second beam is sloped, and is cut to create a gap between the beam and the column.

Before you start

Create the following parts:

- Concrete column (round or rectangular profile)
- Two concrete beams (rectangular, HI, I, L, or inverted T profile)

Selection order

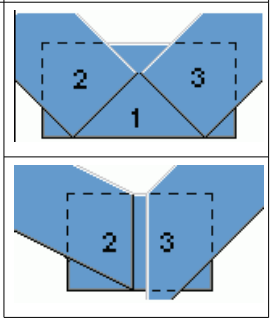
1. Select the main part (column).
2. Select the secondary part (beam).
3. Select the second secondary part (beam).
4. Click the middle mouse button to create the connection.

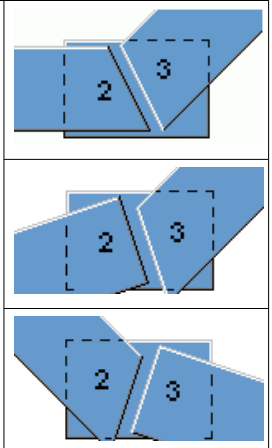
Picture tab

Use the **Picture** tab to define dimensions and the position of the anchor bolts relative to the beams and bearing plates.

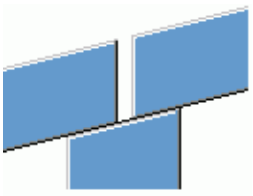
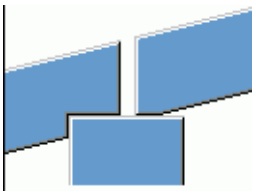
Beam and column

Select one of the following options to define how the end of the beams are cut and shaped:

Key	Options
1 = column 2 = first beam picked 3 = second beam picked	

Key	Options
	

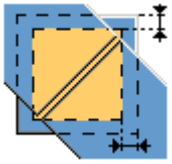
Select one of the following options to fit the beams or the column:

Option	Description
	Fits column. Default
	Fits beam.

Bearing plate

Select one of the following options to define the position of the bearing plate:

Option	Description
	Square with beam. Default
	Square with column.

Option	Description
	Square with column. Enter the distances from column edges.

Dowel tab
Use the **Dowel** tab to define dowel properties.

Option	Description
Profile	Select the dowel profile from the profile catalog.
Prefix, **Start number**	Prefix and start number for the part position number.
Material	Material grade. The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options** .
Name	Name that is shown in drawings and reports.
Finish	Describes how the part surface has been treated.
Class	Enter a number to group the parts that the component creates. By default, the class number affects the color in which the part is shown in model views.
Size	Diameter of the bars.
Grade	Strength of the steel used in the bars.
Number of bars	Select **1 Dowel** to create one reinforcing bar. Select **2 Dowels** to create two reinforcing bars. Then define the distance between the bars in the **Bar distance** field.

Parts tab
Use the **Parts** tab to define bearing pad, drainage hole, grout, bolt plate, nut, and tube properties.

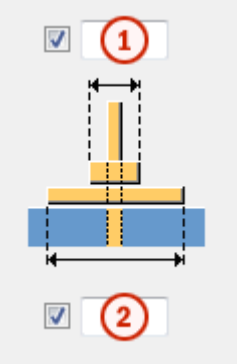
Option	Description
t, **b**, **h**	Define the part thickness, width and height.
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name for the part.
Class	Use **Class** to group the parts.

Left beam/Right beam tab
Use the **Left beam/Right beam** tab to define anchor bolt, bolt hole and recess properties.

Left beam is the first beam selected and right is the second.

You can create a clearance between beam and column if beam is sloped.

Nut and bolt plate



Field	Description
1	Nut width.
2	Bolt plate width.

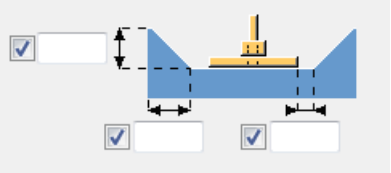
Nut and washer plate

Select one of the following options to define if the nut and washer plate are recessed into the beam:

Option	Description
	Nut and washer plate on the surface of the beam. Default

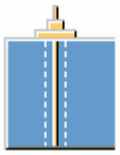
Option	Description
	Nut and bolt plate recessed into the beam.

If you choose to recess the nut and washer plate into the beam, enter the following dimensions to define the recess:



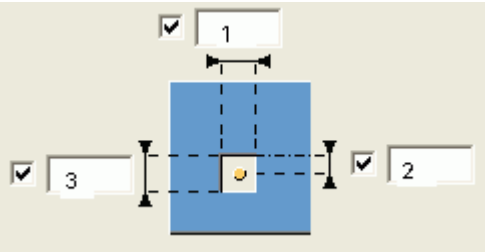
Grout

Select one of the following options to include and define grout:

Option	Description
	No grout. Default
	Bolt hole grouted. No nut or washer plate.
	Bolt hole grouted. Bolt, washer plate, and anchor bolt protruding.

Bolt hole

Enter the following bolt hole dimensions:



Field	Description
1	Hole dimension in the direction of the beam.

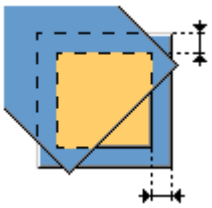
Field	Description
2	The distance from beam center line to hole center and bolt.
3	Hole dimension in the direction perpendicular to the beam.

Select the shape of the bolt hole:

Option	Description
	Rectangular Default
	Round

Bearing plate

Select one of the following options to define the position of the bearing plate:

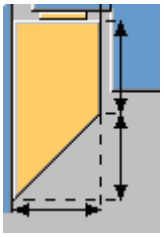
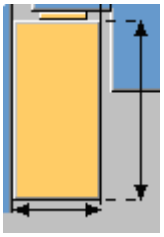
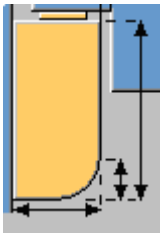
Option	Description
	Square with beam. Default
	Square with column.
	Square with column. Enter the distances from column edges.

Left corbel/Right corbel tab

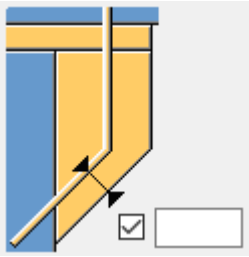
Use the **Left corbel/Right corbel** tab to create a corbel and define its properties.

Select to create a corbel from the **Create corbel** list.

The options for chamfering corbels are:

Option	Description
	Beveled Default
	Straight
	Rounded

Define the placement of the corbel rebar.



Option	Description
t, b, h	Define the thickness, width, and height of the part.
Pos_No	Prefix and start number for the part position number. The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options** .

Option	Description
Name	Name that is shown in drawings and reports.
Class	Part class number.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

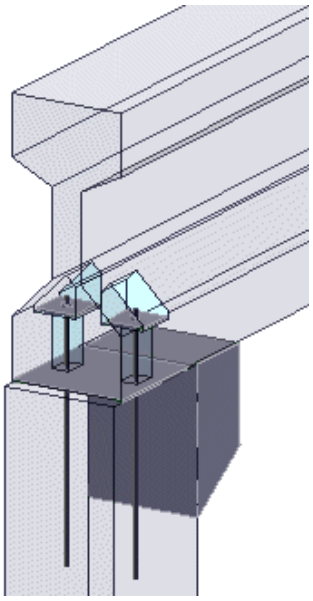
Seating with dowel to flange (77)

Seating with dowel to flange (77) connects the flanges of a beam to a column using anchor bolts and an optional corbel.

Parts created

- Anchor bolts (2)
- Nuts (2)
- Washer plates (2)
- Bearing plate (1)
- Fittings for beam and column
- Holes for bolts (2)
- Corbel (optional)
- Recess for nut and washer plates

Use for

Situation	More information
	Connects the flanges of a beam to a column using anchor bolts and a beveled corbel. Nuts and washer plates recessed.

Before you start

Create the following parts:

- Concrete column (Round or rectangular profile)
- Concrete beam with flange (rectangular, HI, I, L, or inverted T profile)

Selection order

1. Select the main part (column).
2. Select the secondary part (beam).

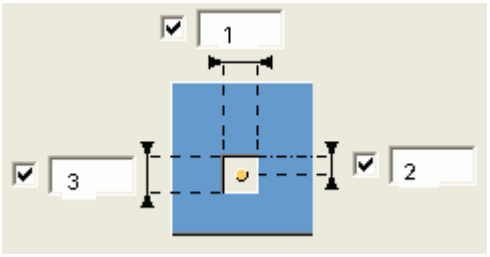
   The connection is created automatically when the second part is selected.

Picture tab

Use the **Picture** tab to define position and length of anchor bolt, and bolt hole dimensions and position.

Bolt hole

Enter the following bolt hole dimensions:



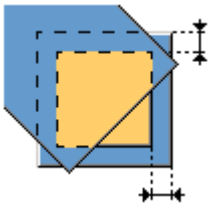
	Description
1	Hole dimension in the direction of the beam.
2	The distance from beam center line to hole center and bolt.
3	Hole dimension in the direction perpendicular to the beam.

Select the shape of the bolt hole:

Option	Description
	Rectangular Default
	Round

Bearing plate

Select one of the following options to define the position of the bearing plate:

Option	Description
	Square with beam. Default
	Square with column.
	Square with column. Enter the distances from column edges.

Dowel tab

Use the **Dowel** tab to define dowel properties.

Select to create the dowel as a reinforcement bar, poly-profile, or custom component part from the **Type** list.

Option	Description
Profile	Select the dowel profile from the profile catalog.
Prefix, **Start number**	Prefix and start number for the part position number.
Material	Material grade. The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options** .
Name	Name that is shown in drawings and reports.
Finish	Describes how the part surface has been treated.
Class	Enter a number to group the parts that the component creates. By default, the class number affects the color in which the part is shown in model views.
Size	Diameter of the bars.
Grade	Strength of the steel used in the bars.

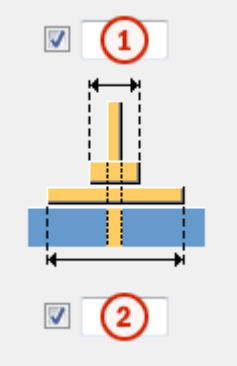
Parts tab

Use the **Parts** tab to define bearing pad, drainage hole, bolt plate, nut and recess properties.

Part properties

Option	Description
t, **b**, **h**	Define the part thickness, width and height.
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name for the part.
Class	Use **Class** to group the parts.

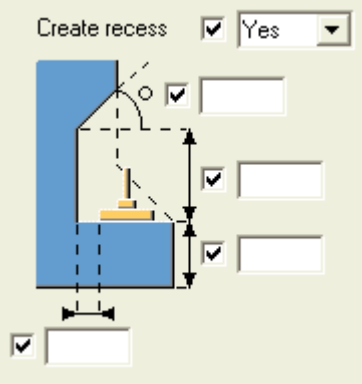
Nut and bolt plate



Field	Description
1	Nut width.
2	Bolt plate width.

Recess

To cut a recess in the beam web, select the **Yes** option in the **Create recess** list. Enter the following dimensions to define the recess:

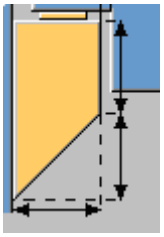
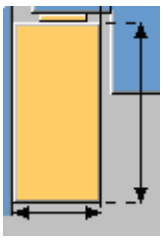
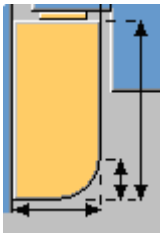


Corbel tab

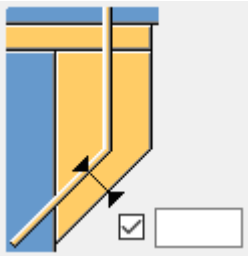
Use the **Corbel** tab to create a corbel and define its properties.

Select to create a corbel from the **Create corbel** list.

The options for chamfering corbels are:

Option	Description
	Beveled Default
	Straight
	Rounded

Define the placement of the corbel rebar.



Option	Description
t, b, h	Define the thickness, width, and height of the part.
Pos_No	Prefix and start number for the part position number. The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options** .
Name	Name that is shown in drawings and reports.
Class	Part class number.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

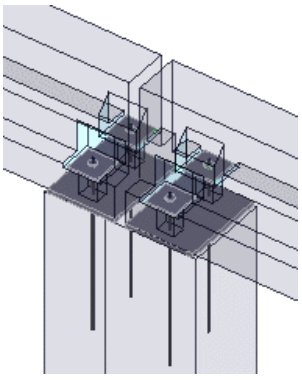
Two-sided seating with dowel to flange (78)

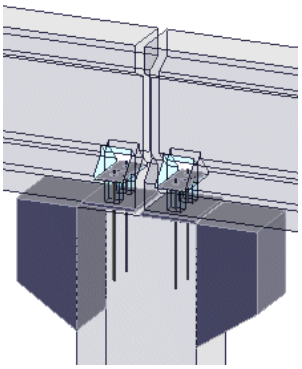
Two-sided seating with dowel to flange (78) connects the flanges of two beams to a column using anchor bolts and optional corbels.

Parts created

- Anchor bolts (4)
- Nuts (4)
- Washer plates (4)
- Bearing plate (2)
- Fittings for beam and column
- Holes for bolts (4)
- Corbels (2) optional
- Recesses for nut and washer plate (4)

Use for

Situation	Description
	Connects the flanges of two beams to a column using anchor bolts.

Situation	Description
	Connects the flanges of two beams to a column using anchor bolts and beveled corbels.

Before you start

Create the following parts:

- Concrete column (Round or rectangular profile)
- Concrete beam with flange (rectangular, HI, I, L, or inverted T profile)

Selection order

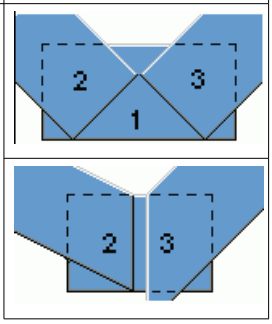
1. Select the main part (column).
2. Select the secondary part (beam).
3. Select the second secondary part (beam).
4. Click the middle mouse button to create the connection.

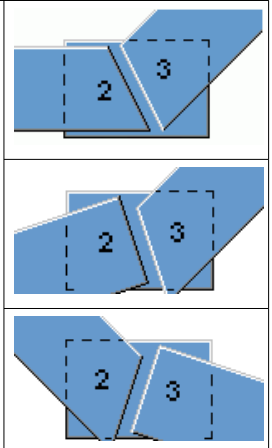
Picture tab

Use the **Picture** tab to define dimensions and the position of the anchor bolts relative to the beam and bearing plate, length of anchor bolts, and beam ends.

Beam and column

Select one of the following options to define how the end of the beams are cut and shaped:

Key	Options
1 = column 2 = first beam picked 3 = second beam picked	

Key	Options
	

Bearing plate

Select one of the following options to define the position of the bearing plate:

Option	Description
	Square with beam. Default
	Square with column.
	Square with column. Enter the distances from column edges.

Dowel tab

Use the **Dowel** tab to define dowel properties.

Select to create the dowel as a reinforcement bar, poly-profile, or custom component part from the **Type** list.

Option	Description
Profile	Select the dowel profile from the profile catalog.
Prefix, **Start number**	Prefix and start number for the part position number.

Option	Description
Material	Material grade. The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options** .
Name	Name that is shown in drawings and reports.
Finish	Describes how the part surface has been treated.
Class	Enter a number to group the parts that the component creates. By default, the class number affects the color in which the part is shown in model views.
Size	Diameter of the bars.
Grade	Strength of the steel used in the bars.

Parts tab

Use the **Parts** tab to define bearing pad, drainage hole, bolt plate, and nut properties.

Option	Description
t, **b**, **h**	Define the part thickness, width and height.
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name for the part.
Class	Use **Class** to group the parts.

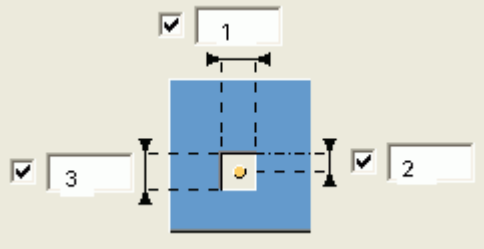
Left beam/Right beam tab

Use the **Left beam/Right beam** tab to define anchor bolt, bolt hole and recess properties.

Left beam is the first beam picked and right is the second.

Bolt hole

Enter the following bolt hole dimensions:

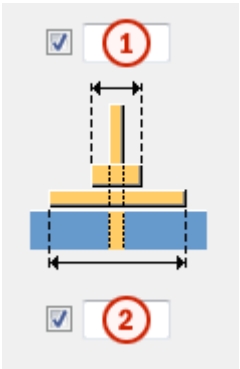


Field	Description
1	Hole dimension in the direction of the beam.
2	The distance from beam center line to hole center and bolt.
3	Hole dimension in the direction perpendicular to the beam.

Select the shape of the bolt hole:

Option	Description
	Rectangular Default
	Round

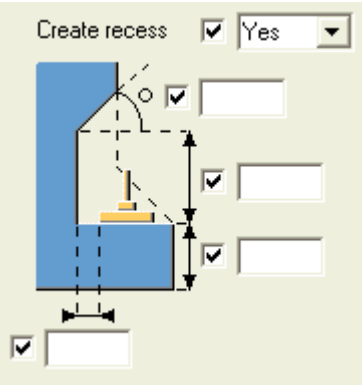
Nut and bolt plate



Field	Description
1	Nut width.
2	Bolt plate width.

Recess

To cut a recess in the beam web, select the **Yes** option in the **Create recess** list. Enter the following dimensions to define the recess:

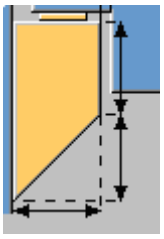
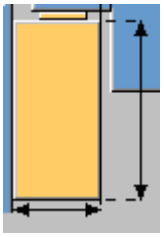
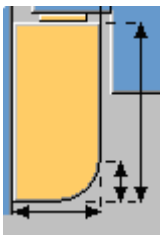


Left corbel/Right corbel tab

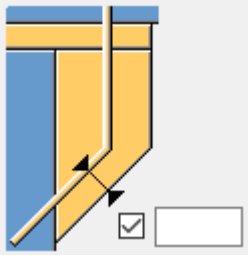
Use the **Left corbel/Right corbel** tab to create a corbel and define its properties.

Select to create a corbel from the **Create corbel** list.

The options for chamfering corbels are:

Option	Description
	Beveled Default
	Straight
	Rounded

Define the placement of the corbel rebar.



Option	Description
t, b, h	Define the thickness, width, and height of the part.
Pos_No	Prefix and start number for the part position number. The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options** .
Name	Name that is shown in drawings and reports.
Class	Part class number.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Beam and column connections

This section introduces components that can be used in concrete connections.

Click the links below to find out more:

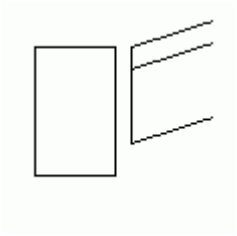
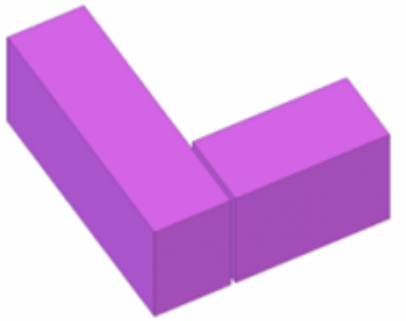
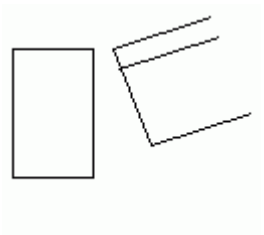
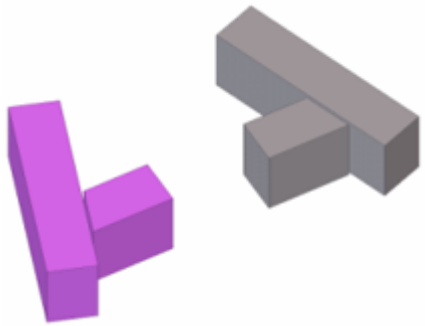
- Battering connection (13) (page 3349)
- Corbel connection (14) (page 3350)
- Column - beam (14) (page 3362)
- Tapered I beam (81) (page 3368)
- Corbels and recesses (82) (page 3377)

- Concrete console (110) (page 3383)
- Concrete console (111) (page 3397)
- Concrete beam-beam (112) (page 3407)

Battering connection (13)

Battering connection (13) fits the secondary part, for example, in a custom component either square to the main part, or at an angle.

Use for

Situation	Description
 	Secondary is fit square to the main part.
 	Secondary is fit to the main part at an angle.

Before you start

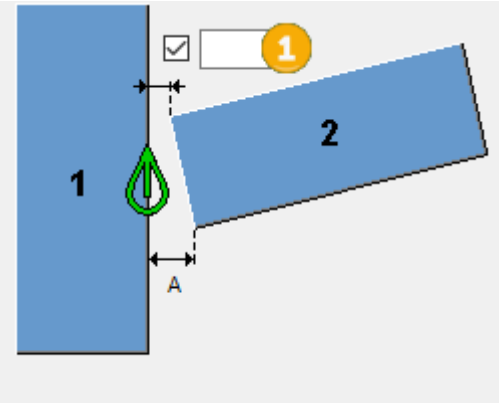
Create two parts.

Selection order

1. Select the main part
2. Select the secondary part.

Picture tab

Use the **Picture** tab to define the angle and the distance between the parts.



	Description
1	Define the angle between the parts. If the part angle makes the gap larger than the value **A**, then the part is cut at the end.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Corbel connection (14)

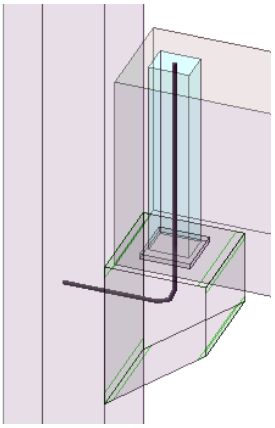
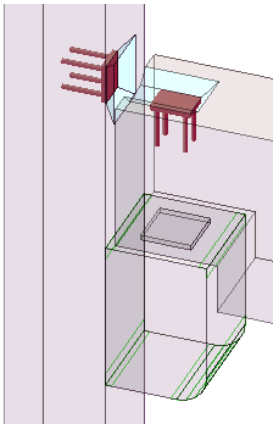
Corbel connection (14) connects a beam to a column using a straight, beveled, or rounded corbel and reinforcing bars or fastening plates.

Objects created

- Corbel

- Bearing plate
- Drainage holes in bearing plate (1 or 2) (optional)
- Reinforcing bars (1 or 2) (optional)
  - Bolt plates for reinforcing bars
  - Nuts for reinforcing bars
  - Recesses for bolt plates and nuts
- Fastening plates (2) (optional)
  - Recesses for fastening plates

Use for

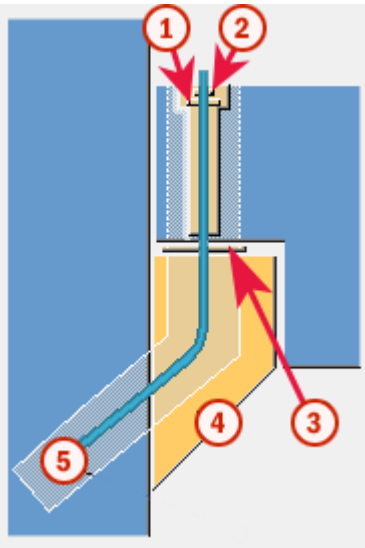
Situation	Description
	Connects a beam to a column using a reinforcing bar and a beveled corbel.
	Connects a beam to a column using fastening plates and a rounded corbel.

Selection order

1. Select the main part (column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

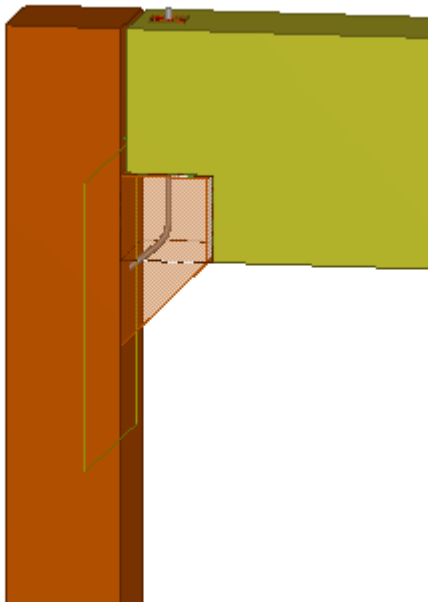


	Part
1	Bolt plate
2	Nut
3	Bearing plate
4	Corbel
5	Reinforcing bar

Example: Add a corbel connection using Corbel connection (14)
In this example, you will add a corbel connection between a column and a beam.

1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. Enter `corbel` in the search box.
3. Select **Corbel connection (14)**.
4. Select the main part (column).
5. Select the secondary part (beam).

Tekla Structures automatically adds the corbel connection between the column and the beam when you select the beam.



Picture tab

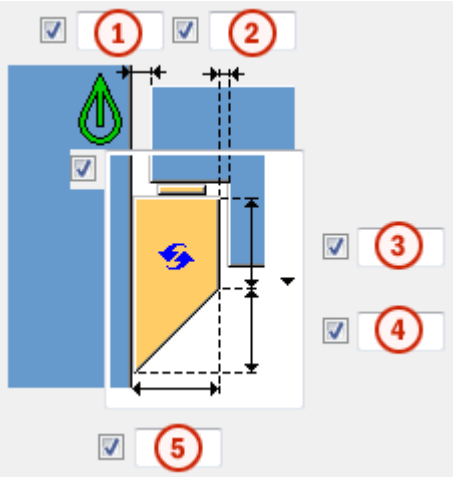
Use the **Picture** tab to control the corbel shape and dimensions, beam end shape and dimensions, and corbel side chamfers in **Corbel connection (14)**.

When beam sloped, adjust

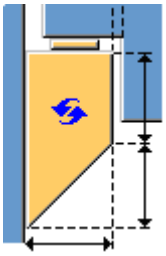
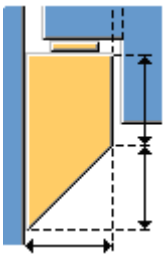
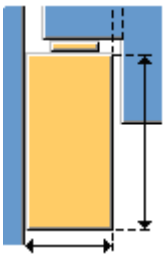
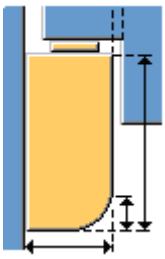
If the beam in the connection is sloped, define whether the beam or the corbel is cut.

When the beam end is cut, the corbel top stays horizontal. When the corbel is cut, the corbel top has the same slope as the beam.

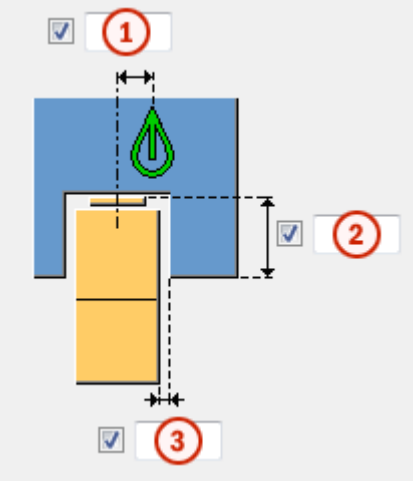
Corbel dimensions and shape



	Description
1	Gap between the column and the beam.
2	Gap between the corbel and the beam.
3	Vertical corbel dimension.
4	Beveled corbel dimension.
5	Corbel width.

Option	Description
	Default Beveled corbel AutoDefaults can change this option.
	Beveled corbel
	Square corbel
	Rounded corbel

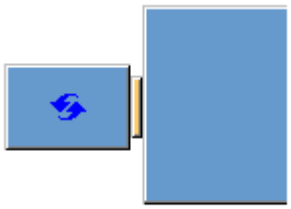
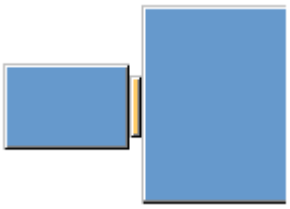
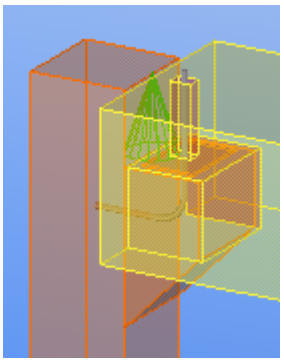
Corbel position

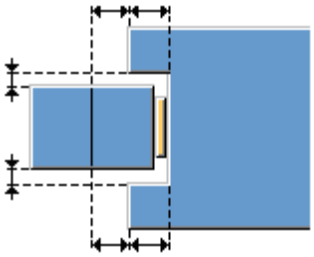
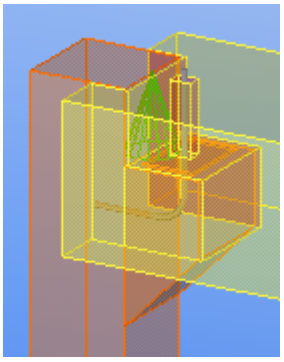
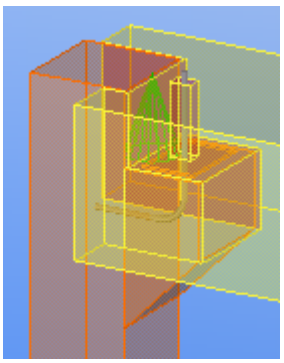


	Description
1	Horizontal corbel offset.
2	Height of the beam cut.
3	Gap between the corbel and extended beam.

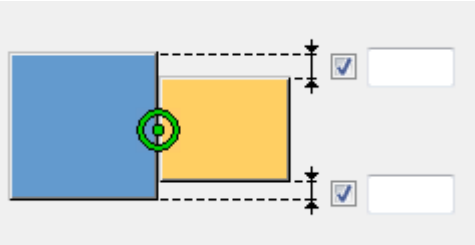
Beam end shape

You can fit the beam end to the column, or you can create a straight beam end.

Option	Description	Example
	Default Straight beam end AutoDefaults can change this option.	
	Straight beam end	

Option	Description	Example
	Beam end is fit to the column based on the dimensions you define. Beam can extend symmetrically on both sides of the column, or with different dimensions.	 

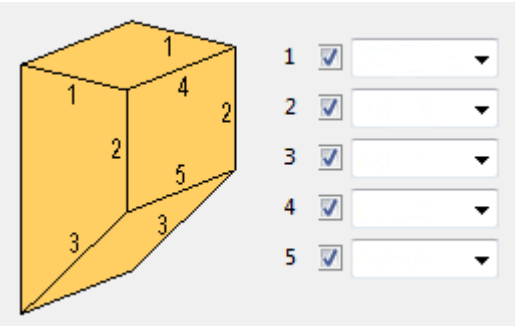
Corbel thickness



Define the distances from column edges to set the corbel thickness.

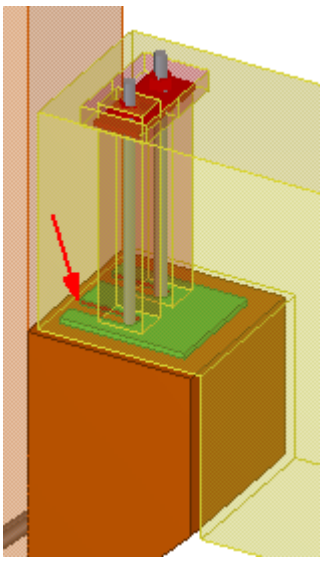
Corbel side chamfers

Define whether the corbel sides are chamfered. By default, no chamfers are created.



Parts tab
Use the **Parts** tab to control the corbel part properties and dimensions in **Corbel connection (14)**.

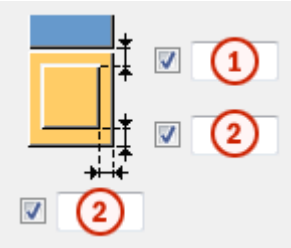
Corbel parts

Part	Description
Bearing plate	Bearing plate thickness.
Cast unit	Select whether cast unit is formed.
Drainage hole	Select whether drainage holes for each reinforcing bar are created in the bearing plate. 
Bolt plate	Bolt plate thickness.
Nut	Nut thickness.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined on the **Components** tab in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box on the **Components** tab in **File**

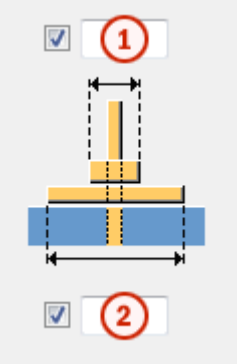
Option	Description	Default
		menu --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	

Bearing plate dimensions



	Description
1	Bearing plate distance from the column edge.
2	Bearing plate distance from the corbel edges.

Nut and bolt plate dimensions



	Description
1	Nut width.
2	Bolt plate width.

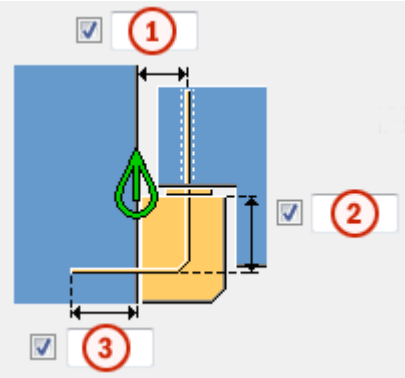
Reinforcing bar tab

Use the **Reinforcing bar**tab to control the reinforcing bar and fastening plate properties, and the bolt plate and fastening plate recesses in **Corbel connection (14)**.

Reinforcing bar properties

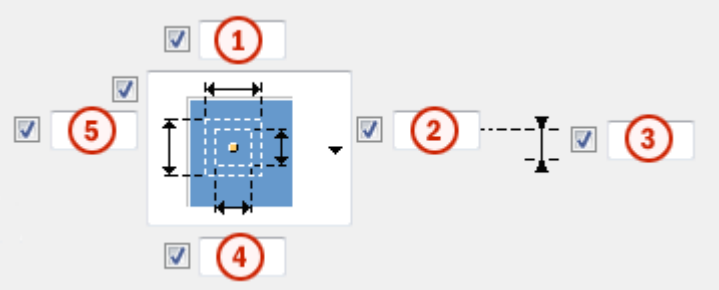
Option	Description
Grade	Strength of the steel used in the reinforcing bars.
Size	Diameter of the reinforcing bar.
Bending radius	Internal radius of the bends in the bar.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcements. For example, you can display different reinforcement classes in different colors.
Number of bars	Select **1 Dowel** to create one reinforcing bar. Select **2 Dowels** to create two reinforcing bars. Then define the distance between the bars in the **Bar distance** field.

Reinforcing bar length



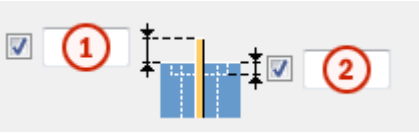
	Description
1	Distance between reinforcing bar center line and the column edge.
2	Vertical length of the reinforcing bar inside the corbel.
3	Length of the reinforcing bar inside the column.

Recess for bolt plate and nut



	Description
1	Size of the bolt plate recess in the x direction.
5	Size of the bolt plate recess in the y direction.
2	Size of the nut recess in the y direction.
4	Size of the nut recess in the x direction.
3	Reinforcing bar offset.

Reinforcing bar extra length



	Description
1	Extra length of the reinforcing bar.
2	Length of the reinforcing bar inside the recess.

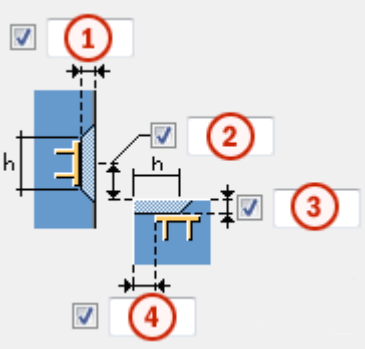
Connecting devices

Define the connecting devices that connect the beam and the column.

Option	Description
	Default One or two reinforcing bars bent to the same angle as the corbel bevel Available for bevel corbels. AutoDefaults can change this option.
	One or two reinforcing bars bent to the same angle as the corbel bevel Available for bevel corbels.

Option	Description
	One or two reinforcing bars Default for straight and rounded corbels.
	Two fastening plates Use custom components as fastening plates.

Recess for fastening plate



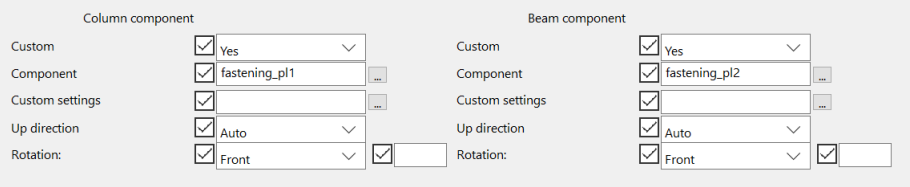
	Description
1	Depth of column recess.
2	Depth of beam recess.
3	Offset of column recess.
4	Offset of beam recess.
Column recess **Beam recess**	Height and width of column and beam recess.

Using custom components as fastening plates

You can use custom components as fastening plates. Use the **Column component** and **Beam component** sections to define the fastening plates in the column and beam.

1. Select the following option in the **Connecting devices** list:

2. In the **Custom** list, select **Yes**.



3. Click the **...** button next to the **Component** field to open the **Select component** dialog box.
4. Browse for the custom component you want to use as fastening plate.

   The component you select must be a custom part and have two or more input points.
5. Select the component and click **OK**.
6. To use saved custom component properties, select the saved properties file in **Custom settings**.
7. If the direction or rotation of the fastening plate is not correct, select another option in the **Up direction** or **Rotation** list.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

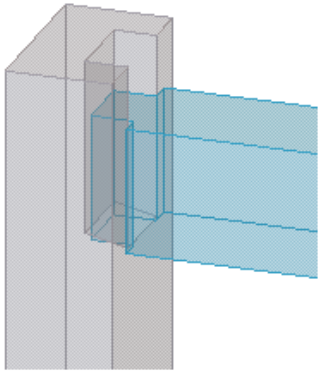
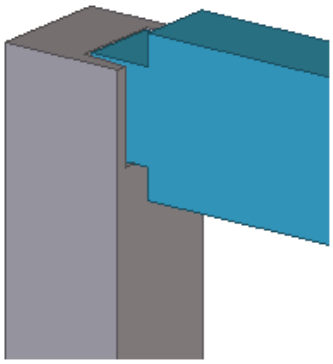
Column - beam (14)

Column - beam (14) creates a connection between a concrete column and a concrete beam or wall/panel. The orientation of the beam can be horizontal or sloped. Note that the connection works only with precast parts.

Objects created

- Cuts
- Fittings

Use for

Option	Description
	Beam end rests on the column.
	Beam end rests on the column.

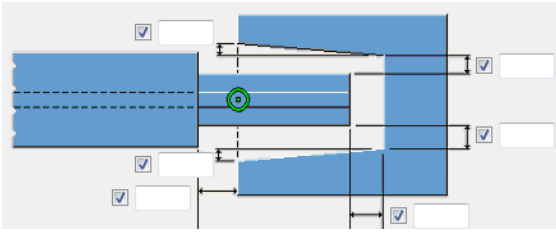
Selection order

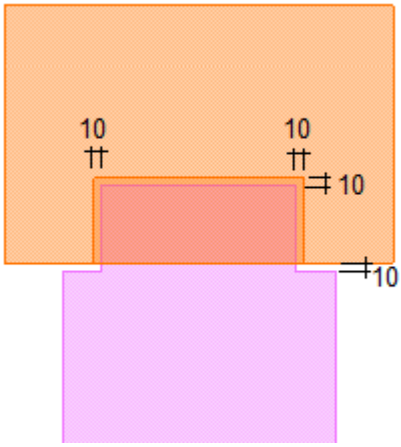
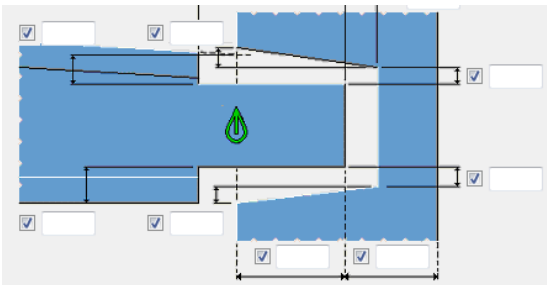
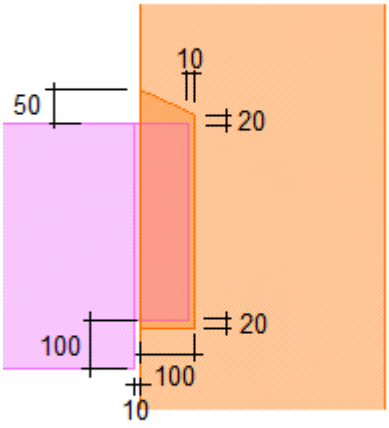
1. Select the main part (column).
2. Select the secondary part (beam/wall/panel).

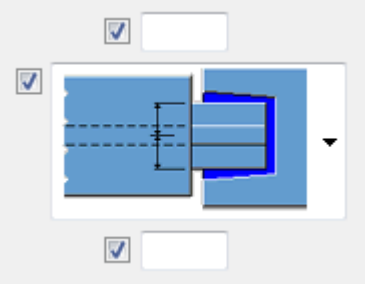
Picture tab

Use the **Picture** tab to define cut-out shapes and cut-out dimensions of the column and the beam.

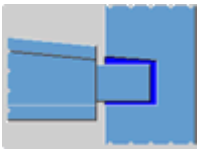
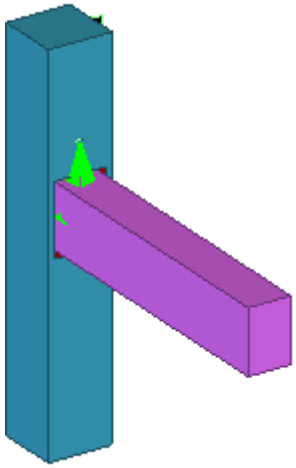
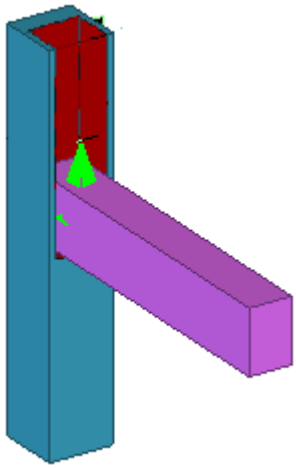
Cut-out dimensions

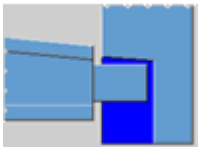
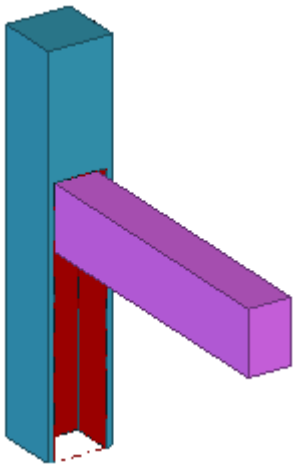
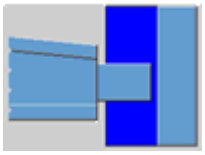
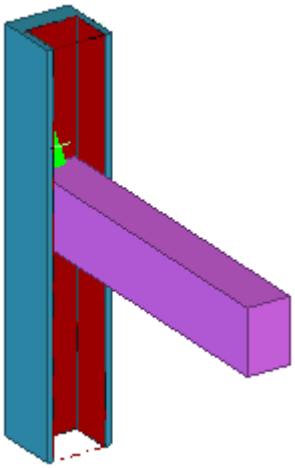
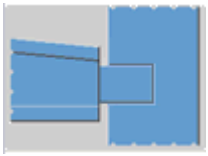
Option	Description
	Cut-outs of the main part and the secondary part in the horizontal direction. The space between the column and the beam can be defined at both sides. For tapered openings

Option	Description
	you can define how much the opening is reduced. Example: 
	Cut-outs of the main part and the secondary part in the vertical direction. The space between the column and the beam can be defined at both sides. For tapered openings you can define how much the opening is reduced. Example: 

Option	Description
	Select the how the opening is reduced. The options are: • From beam centerline • From beam edges

Cut-out shape

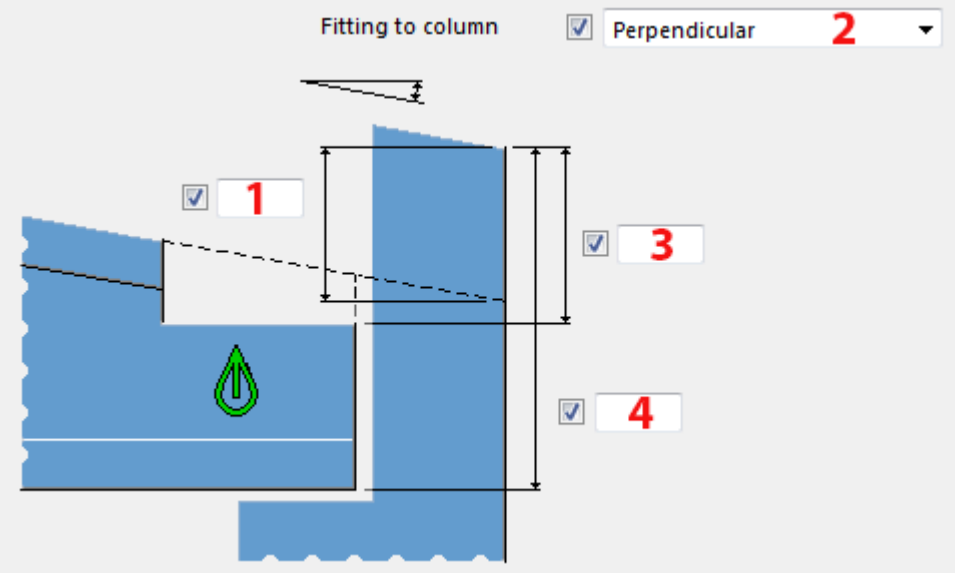
Option	Description
	Cut around the secondary part 
	Cut-out to the top of the column 

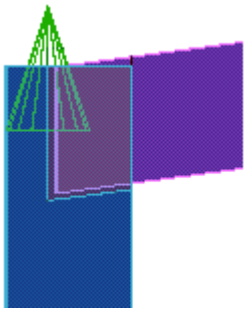
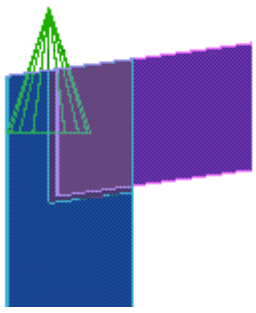
Option	Description
	Cut-out to the bottom of the column 
	Full vertical cut-out 
	No cut-out

Column tab
Use the **Column** tab to define how the top of the column is modified.

Column extension

Define the column extension. If you do not enter any values, the column is extended to its original top level.



	Description
1	Column extension in the vertical direction from the beam top. This dimesion has the highest priority among the three dimensions (**1**, **3**, **4**) for defining the column extension.
2	Select wether the top of the column is perpendicular or parallel to the beam. **Perpendicular**:  **Parallel with top of beam**: 
3	Column extension in the vertical direction.

	Description
4	Column extension in the vertical direction from the beam bottom.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

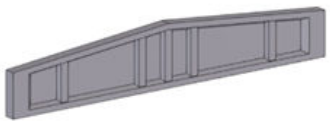
Tapered I beam (81)

Tapered I beam (81) creates beams with multiple types of cross-sections and height variations. Cross-section types include I, T, inverted T, and rectangular. The beam height can be tapered, tapered ridge, or constant.

Objects created

- Beam, one part or multiple parts
- Stiffeners
- End blocks

Use for

Example	Example	Example
		

Before you start

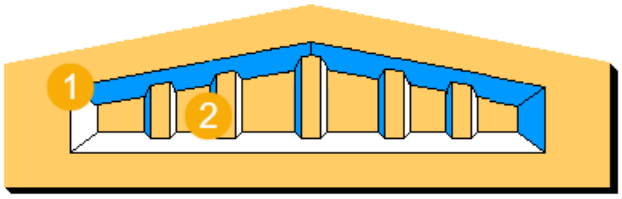
Ensure that you have two points to pick.

Selection order

1. Pick the beam start point.
2. Pick the beam end point.

   The beam is created automatically when you pick the end point.

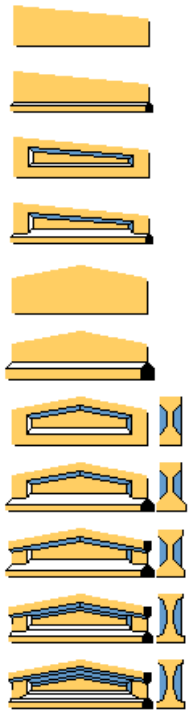
Part identification key

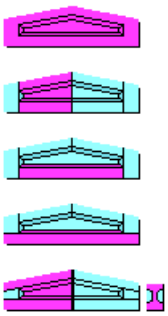


	Description
1	Tapered beam
2	Stiffener

Parameters tab
Use the **Parameters** tab to define the beam type and the dimensions.

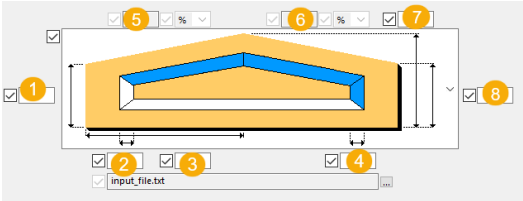
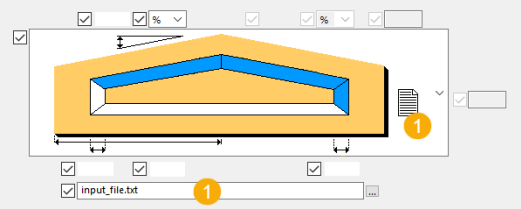
Beam type

Option	Description
	Select the beam shape type.

Option	Description
	Select whether the beam is created as one part or as separate parts. The available options depend on the beam shape type you have selected. The magenta color indicates the main part of the cast unit and the cyan color indicates the secondary part.

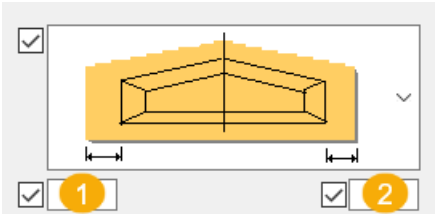
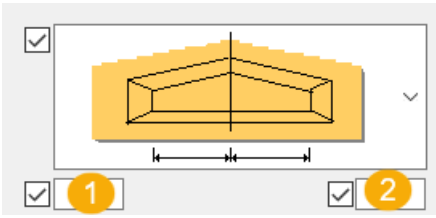
Beam dimensions

The available dimensions depend on the option you select. The example images below show the dimensions you can define.

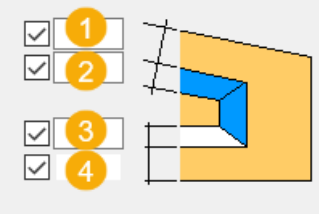
Option	Description
	**1** Height of the beam on the start point side **2** Width of the chamfer on the start point side **3** Location of the ridge. If left empty, the ridge is in the middle. **4** Width of the chamfer on the end point side **5** Angle of the start point edge of the beam, either as percentage or in degrees **6** Angle of the end point edge of the beam, either as a percentage or in degrees **7** Dimension of the beam from the bottom to the highest edge **8** Height of the beam on the end point side
	**1** You can use an external configuration file for defining the dimensions. Define the path to the external file.

End block dimensions

You can define the end block dimensions with certain beam shape types.

Option	Description
	**1** Length of the start point end block **2** Length of the end point end block
	**1** Dimension between the start point end block and the middle of the beam **2** Dimension between the end point end block and the middle of the beam
Position h middle	Select the middle height position: • The height is in the middle of the picked points. Offsets are not taken into account. • The height is in the middle of the actual beam, including the offsets.

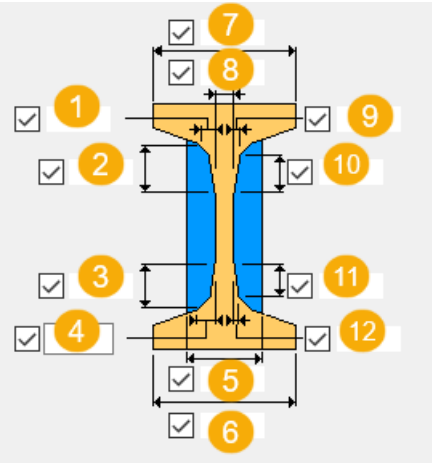
Flange and chamfer dimensions



	Description
1	Height of the top flange
2	Height of the top chamfer
3	Height of the bottom chamfer
4	Height of the bottom flange

Profile cross section dimensions

The available dimension options depend on the selected beam shape type.



	Description
1	Width of the first top chamfer
2	Height of the first top chamfer
3	Height of the first bottom chamfer
4	Width of the first bottom chamfer
5	Width of the end detail
6	Width of the bottom flange
7	Width of the top flange
8	Width of the web
9	Width of the second top chamfer
10	Height of the second top chamfer
11	Height of the second bottom chamfer
12	Width of the second bottom chamfer

Properties tab

Use the **Properties** tab to define the part properties. These settings affect all parts created by the component: beam parts, stiffeners, and end detail parts.

Properties

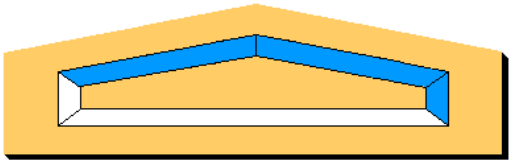
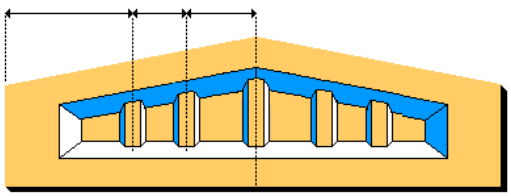
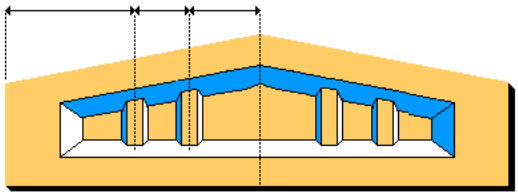
Option	Description
Name	Part name
Material	Material grade
Finish	Part finish
Class	Part class number
Position on plane	Horizontal position of the beam in relation to the input points. You can define an additional offset, if needed.

Option	Description
Position at depth	Depth position of the beam in relation to the input points. You can define an additional offset, if needed.
End offset	End offsets of the part
Cast unit numbering	Numbering prefix and start number for the cast unit
Global displacement	Offset for the part in the x, y, and z direction
Method	Method to connect the created parts together

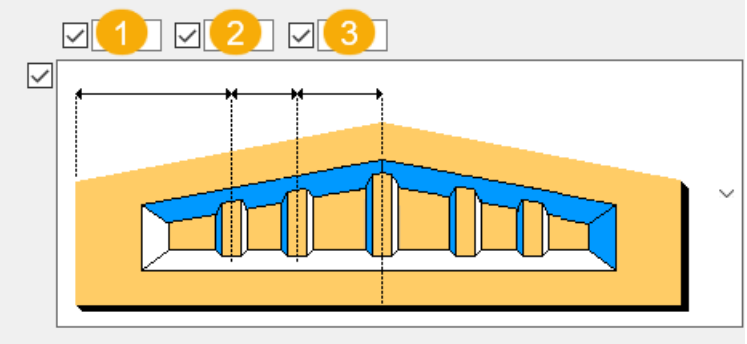
Stiffeners tab

Use the **Stiffeners** tab to define the stiffener creation.

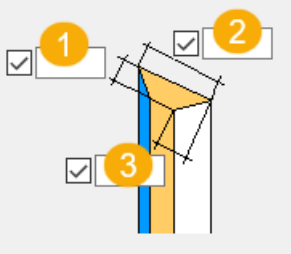
Stiffener type

Option	Description
	Stiffeners are not created.
	Stiffeners are created. The middle stiffener is created.
	Stiffeners are created. The middle stiffener is not created.

Stiffener dimensions



	Description
1	Dimension from the edge of the beam to the first stiffener
2	Dimension from the first stiffener to the second stiffener
3	Dimension from the second stiffener to the central stiffener If the central stiffener is not created, this is the dimension from the second stiffener to the center of the beam. Note that if all three values are defined, only values 1 and 2 will be used.



	Description
1	Thickess of the stiffener
2	Width of the stiffener
3	Width of the top part of the stiffener

Connection method

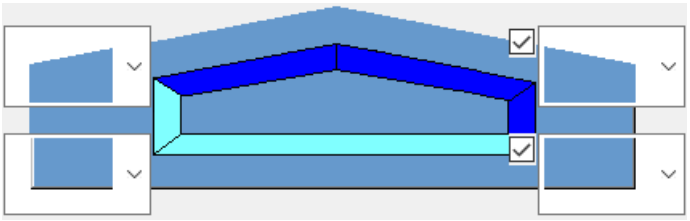
Select how the stiffeners are connected to the main part.

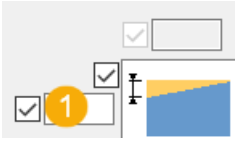
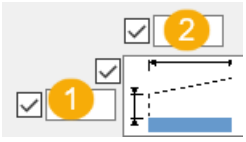
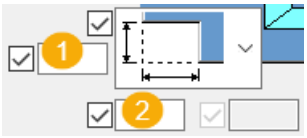
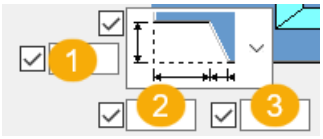
End details tab

Use the **End details** tab to define the end detail creation.

End detail shape and dimensions

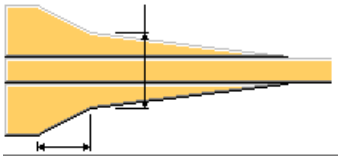
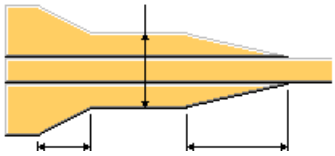
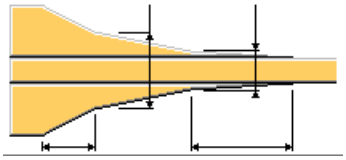
Select the top and bottom end detail on both sides of the beam.



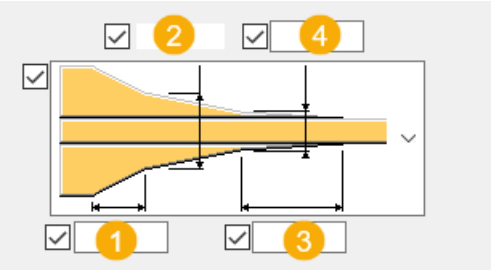
Option	Description
	No end detail at the top
	Horizontal dimension of the added block
	Vertical dimension of the added block
	**1** Vertical dimension of the cut **2** Horizontal dimension of the cut
	No end detail at the bottom
	**1** Vertical dimension of the cut **2** Horizontal dimension of the cut
	**1** Vertical dimension of the cut **2** Horizontal dimension of the cut **3** Horizontal dimension of the first chamfer

End block shape

Option	Description
	End blocks are not created.

Option	Description
	End blocks without chamfers are created.
	End blocks with one chamfer are created.
	End blocks with two chamfers are created.
	End blocks with three chamfers are created.

End block dimensions



	Description
1	Length of the first chamfer
2	Width of the end blocks
3	Width of the end blocks at the end of the second chamfers
4	Width of the third chamfers If there are two chamfers, this is the width of the second chamfer.

UDA tab

Use the **UDA** tab to define the user-defined attributes.

- **String** for text
- **Integer** for integers

- **Float** for values with decimals after a comma
- **Option** for predefined values

Corbels and recesses (82)

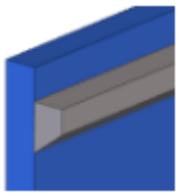
Corbels and recesses (82) adds concrete parts or cuts to a concrete part. You can define the position, offsets and rotation of the added parts or cuts in several ways.

Objects created

The component can add a maximum of four parts or cuts to a concrete part. The added parts can be welded to the main part, added as parts and cast units, or as sub-assemblies.

Use for

- Cut-out seams from concrete walls
- Adding concrete support blocks to concrete columns or walls



Selection order

1. Select the main part.

   The part or cut is created automatically.

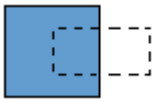
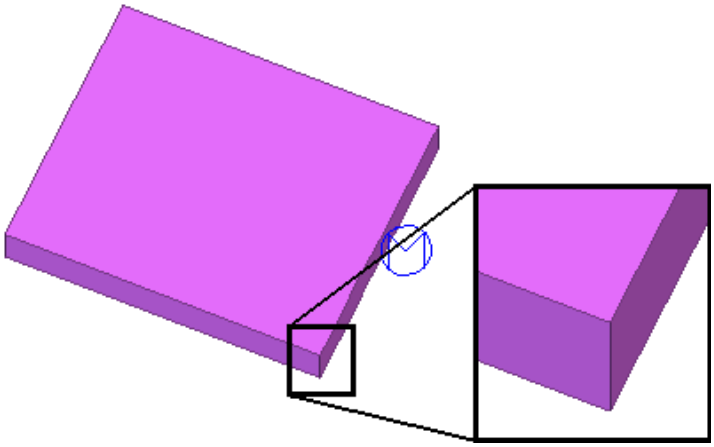
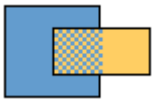
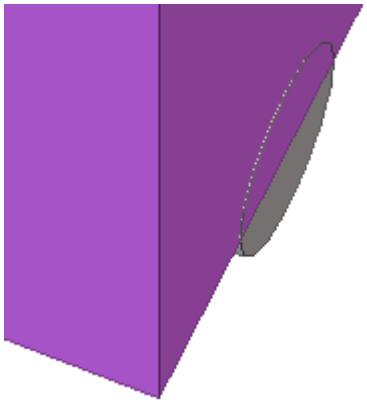
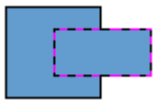
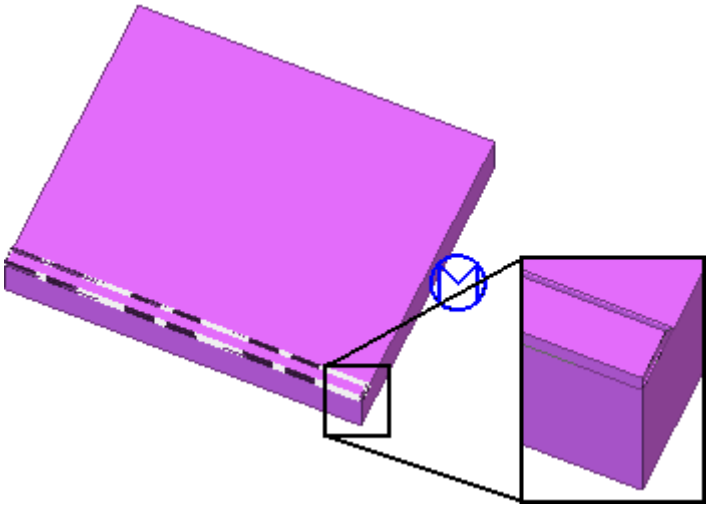
Part 1 / Part 2 / Part 3 / Part 4 tab

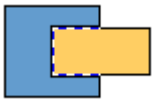
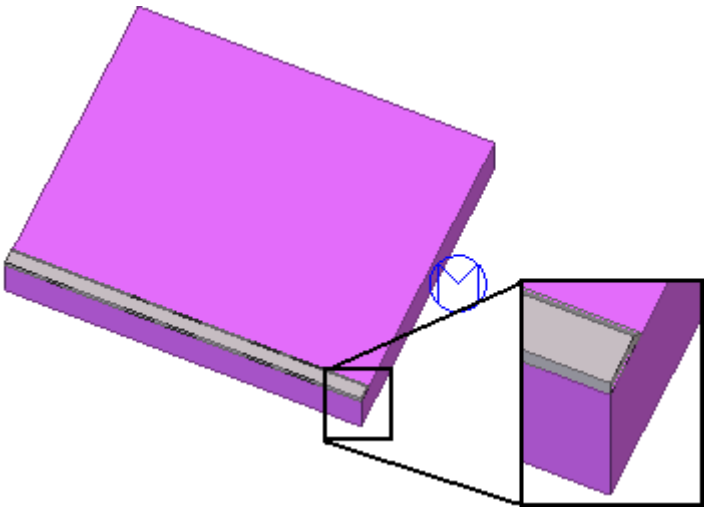
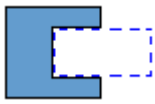
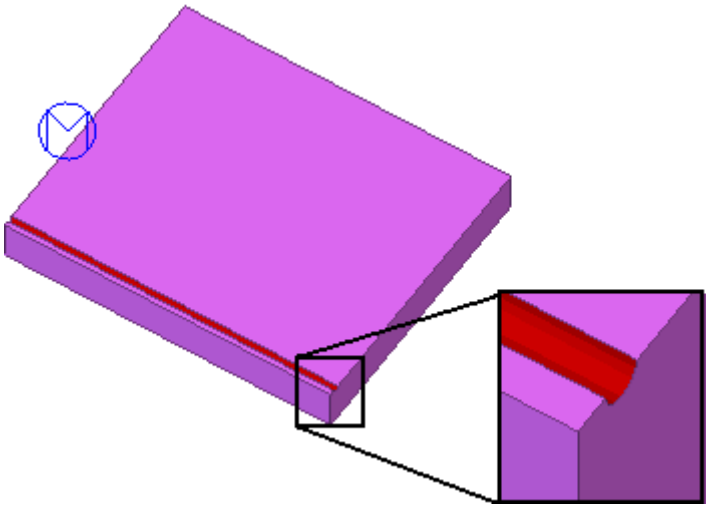
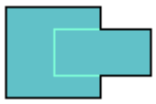
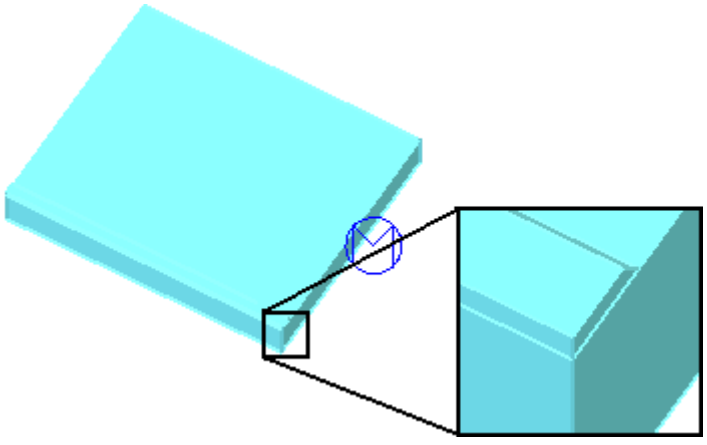
Use the **Part 1**, **Part 2**, **Part 3** or **Part 4** tab to create one additional part or cut to the concrete part.

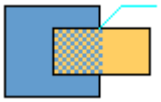
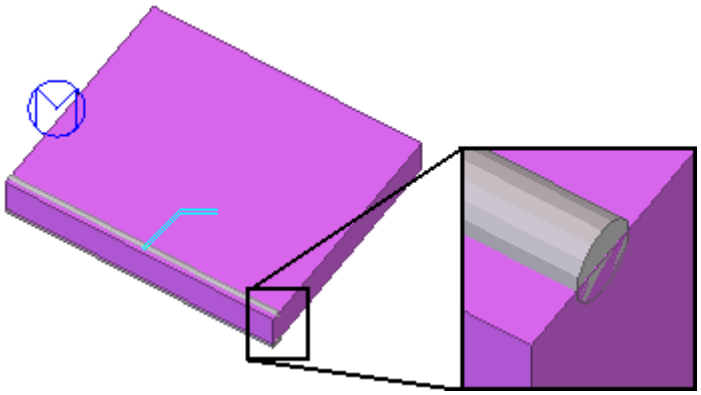
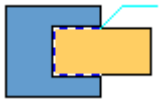
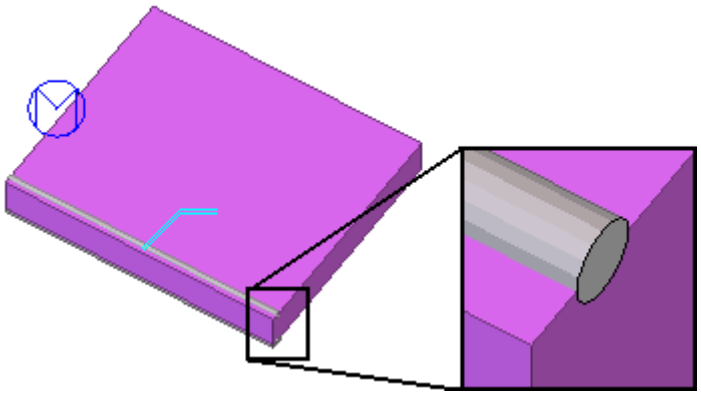
Profile

Select the profile to be added to the concrete part or used for the cut.

Select how the added part is connected to the main part.

Option	Example
	No part created. 
	Part is created. 
	Part is created and added to the main part. 

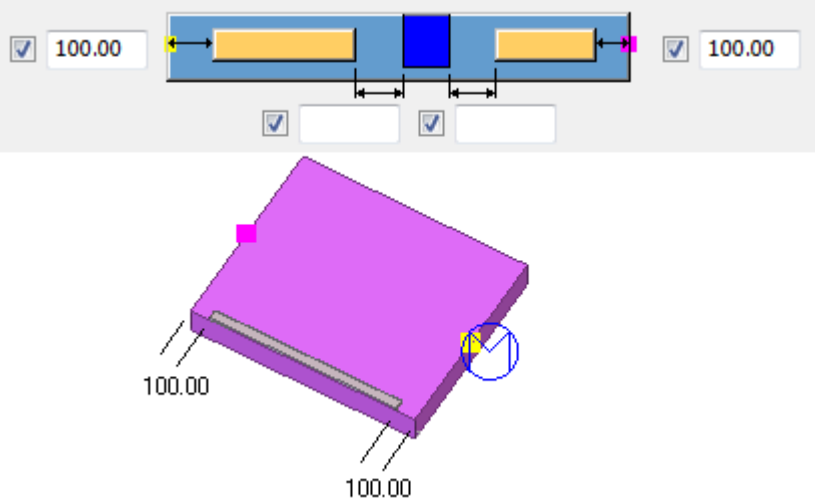
Option	Example
	Part is created and the main part is cut. 
	Main part is cut. 
	Part is created and cast unit is created. 

Option	Example
	Part is created and welded to the main part. 
	Part is created and welded to the main part, and the main part is cut. 
	Part is created and added as sub-assembly.
	Part is created and added as sub-assembly, and the main part is cut.

Offset

Define the offset of the added part from the main part. You can set the offset also for openings.

By default, the added parts or cuts are created between the end points of the concrete part.



Coordinate system

On the **Part 1** tab, select **Use global xy-plane** to place the corbels on the global XY plane or **Use local** to place the corbels on the local XY plane of the part.

Note that setting the coordinate system on the **Part 1** tab also affects the corbels that are created on the **Part 2 - 4** tabs.

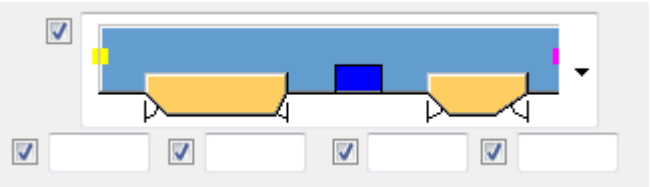
Create diamond corbel around column

Select whether a diamond corbel is created around the column.

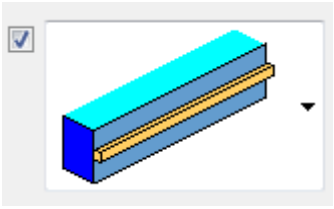
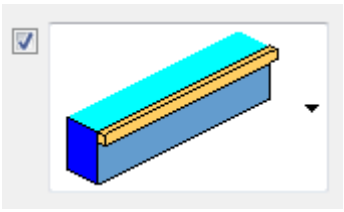
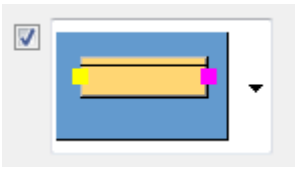
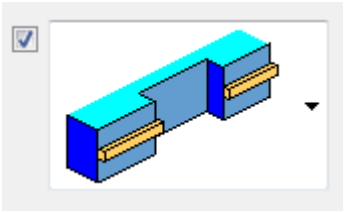
If you create the corbel, you cannot enter any values on the **Part 2**, **Part 3** or **Part 4** tabs.

Chamfers

You can chamfer the added parts. You can define the chamfers either as angles or as dimensions.

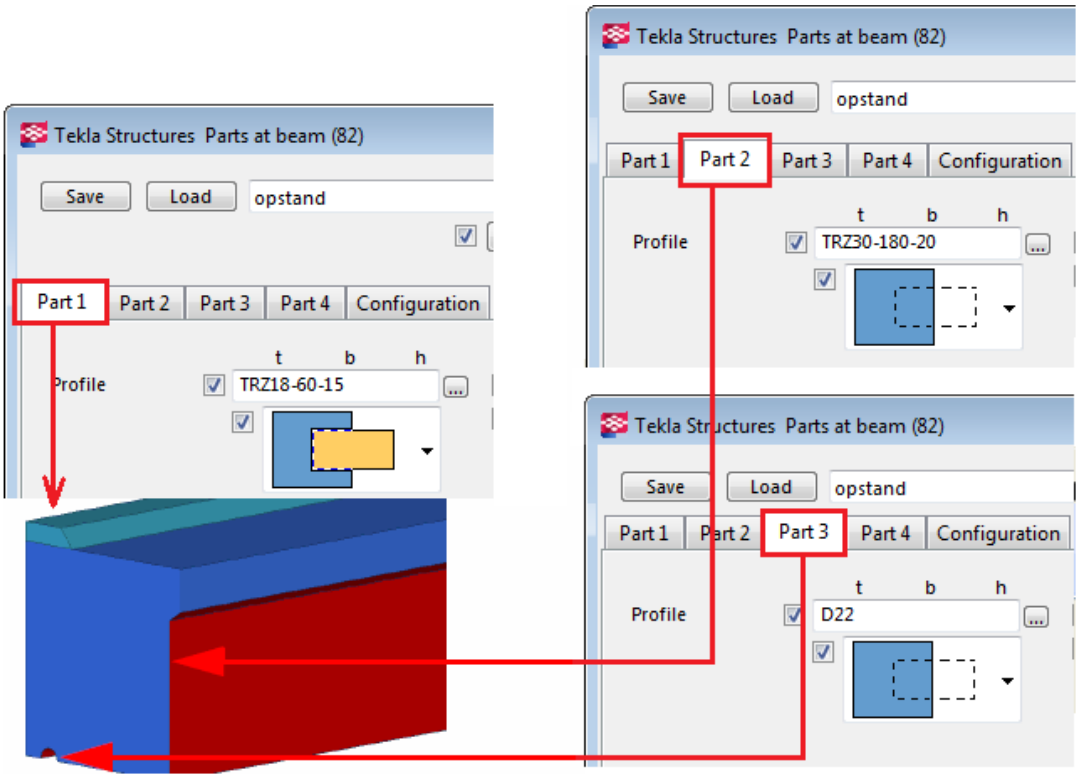


Positioning of the parts

Option	Description
	Select the side where the added parts or cuts are created.
	Select the alignment of the added parts or cuts.
	You can swap the end points of the added parts or cuts.
	Select whether the existing cuts should be taken into account when the added parts are created.

Part 2 / Part 3 / Part 4 tab

You can add or cut more than one part at the same time. To do this, use the tabs **Part 2**, **Part 3** and **Part 4**.



Configuration tab
Use the **Configuration** tab to define the distance in material and whether information needs to be printed.

UDA tab
Use the **UDA** tab to define user-defined attributes for the parts.

Use the **Fabricator name**, **Name**, **Type**, **Nomination**, **Article number** and **Comment** user-defined attributes (UDAs) to add information for the added parts or cuts.

Concrete console (110)

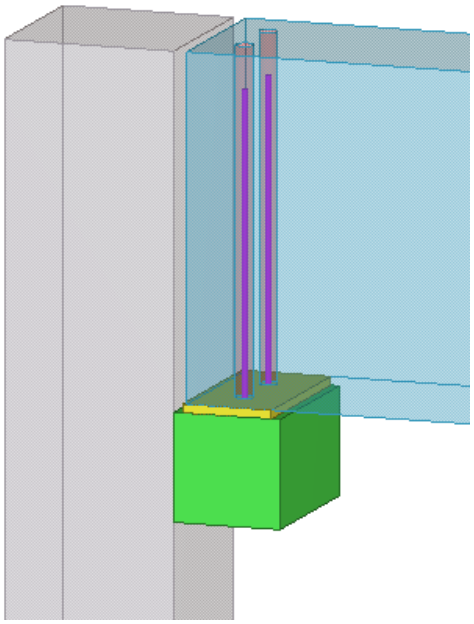
Concrete console (110) creates a connection between a concrete column and a concrete beam. The beam rests on a console which is attached to the column.

Objects created

- Console
- Neoprene strip
- Steel plate between console and beam
- Anchor rods
- Tubes

- Sockets

Use for

Situation	Description
	Console connection between a concrete column and a concrete beam.

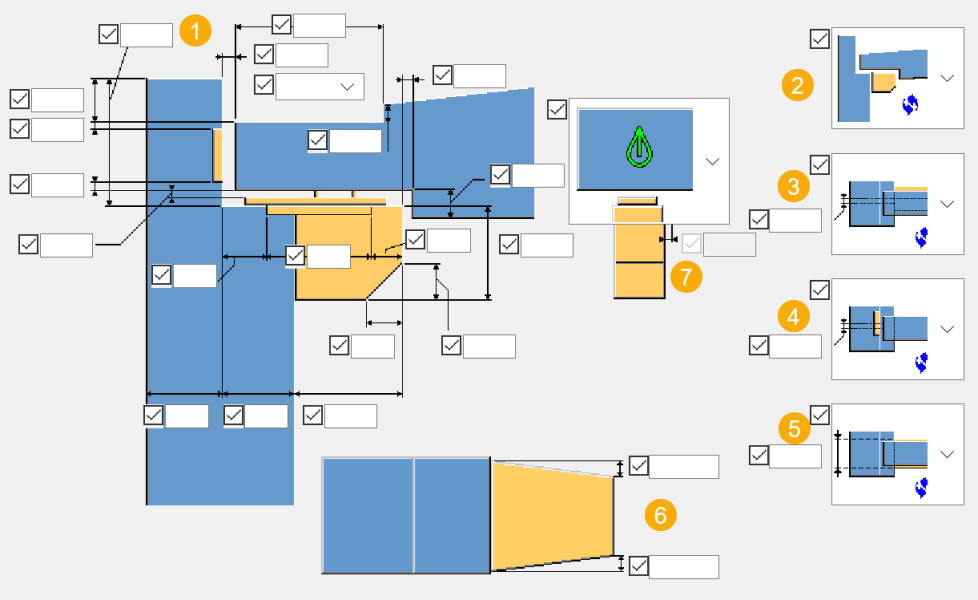
Selection order

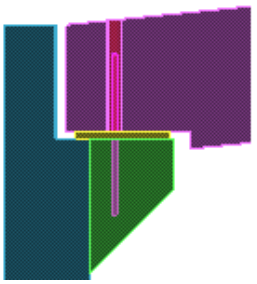
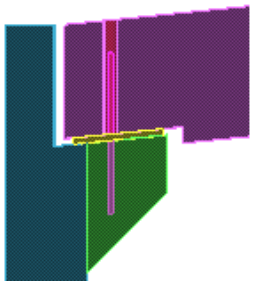
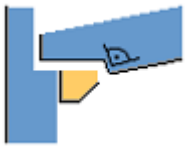
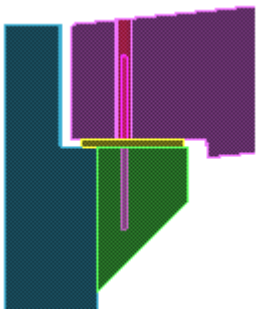
1. Select the main part (column).
2. Select the secondary part (beam).

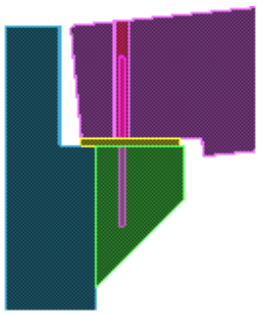
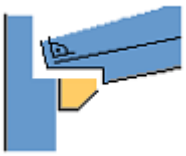
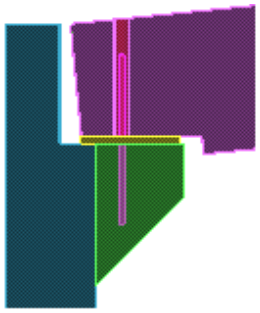
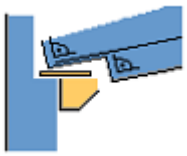
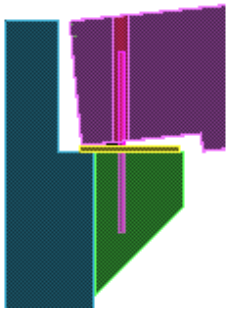
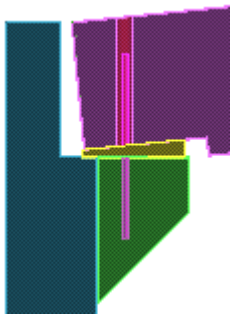
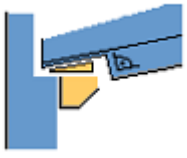
   The connection is created automatically when the secondary part is selected.

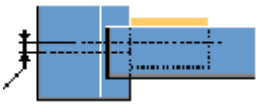
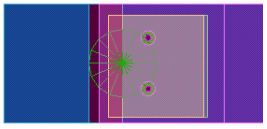
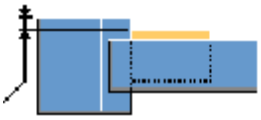
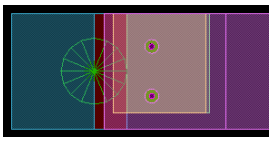
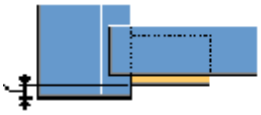
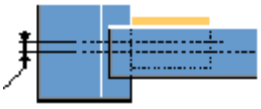
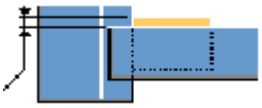
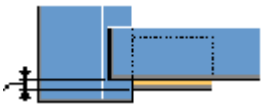
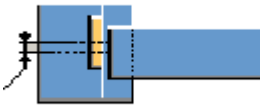
Picture tab

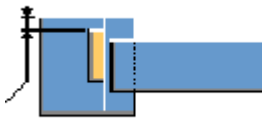
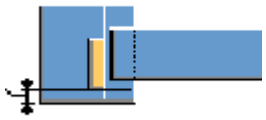
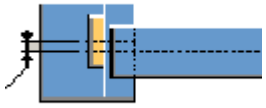
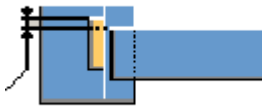
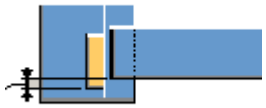
Use the **Picture** tab to define the shape and the dimensions of the console, and the steel support plates and the neoprene layer.



	Description		
1	Shape and dimensions of the concrete console and the steel and neoprene parts.		
2	Select how the shape of the secondary beam is modified if the secondary beam is sloping.		
			
			

	Description		
			
			
			
			
			

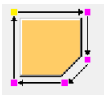
	Description		
3	Console offset. You can set the reference to be the main part or the secondary part. Additionally, the offset distance can be defined.		Reference = column centerline If no offset value is applied, the console is placed symmetrically compared to the column. 
			Reference = column left side 
			Reference = column right side
			Reference = beam centerline
			Reference = beam left side
			Reference = beam right side
4	Vertical plate offset. You can set the reference to be the main part or the secondary part. Additionally, the offset distance can be defined.		Reference = column centerline

	Description		
			Reference = column left side
			Reference = column right side
			Reference = beam centerline
			Reference = beam left side
			Reference = beam right side
5	Define the console thickness. Select whether the console thickness is taken from the secondary part or the main part. The default option is that the console thickness is taken from the secondary part.		
6	Console tapering.		
7	Select whether to create a cut around the console in the secondary part. Define the cut dimension from the edge of the cut to the edge of the console.		

Parts tab

Use the **Parts** tab to define properties for the console, the neoprene layer and the optional steel plates.

Option	Description
Concrete console	Prefix and start number for the part position number, material, name, class and comment for the console.
Console to column	Select how the console is attached to the concrete column. Default is **Part Add**. The **No action** option means that the console is a loose part and not attached to any other part in the component.
Create console like	Select how the console is created. Default is **Contour plate**. **Contour plate** = the console is created using the **Contour plate** command. **Beam** = the console is created using the **Beam** command.
Neoprene	Neoprene layer properties. A neoprene plate for shock absorbing and sound-damping can be created between the beam and the console. If a trapezium shaped neoprene block is used, the defined thickness will be the thickness on the column side.
Neoprene to	Select to which part the neoprene layer is attached, and how the neoprene is attached. Default is **Beam** and **Weld**.
Holes in neoprene	Select how the holes in the neoprene part are created. Default is **By bolt**.
Diameter of holes in neoprene	Diameter of the holes in the neoprene part. By default, the hole size in the neoprene part is equal to the holes in the console. Enter a value to overwrite this default hole size.
Steel plate horizontal	Horizontal steel plate size and properties. The plate is placed under the neoprene part.
Steel plate vertical	Vertical steel plate size and properties. The plate is placed at the console side.
Add steel plate to the column by	Select how the steel plate is attached to the column. Default is **Weld**.

Option	Description
Rotate console polygon	Define the rotation start point of the console polygon. By default, the rotation starts from point `1`. Enter `2`, `3`, `4`, or `5` to change the start point. 
Console polygon direction	Select the direction of the console polygon.

Connection tab

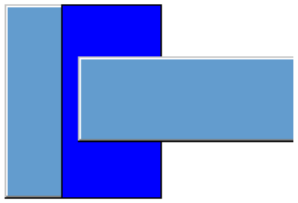
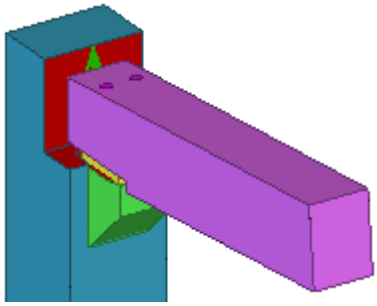
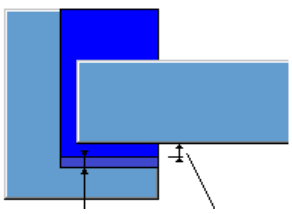
Use the **Connection** tab to define the properties of the anchor rods, nuts and washers and injection tubes and to select how these parts are connected to the console or the main part.

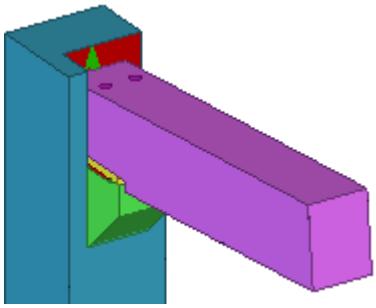
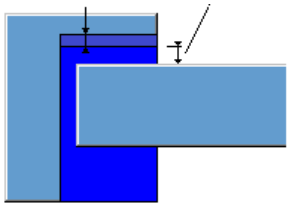
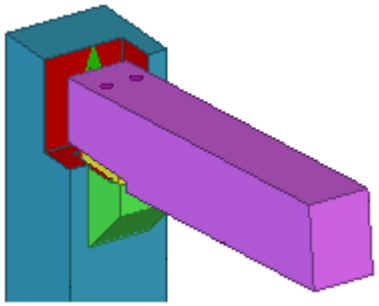
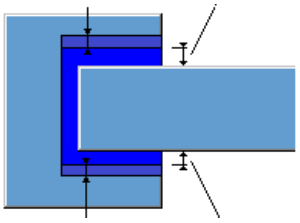
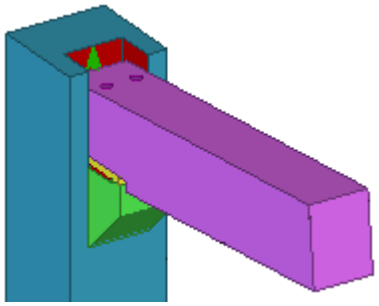
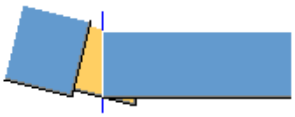
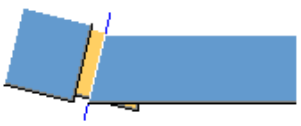
Option	Description
Anchor rods	Anchor rod profile. Define the length and number of rods on the **Anchor rods** tab.
Type of reinforcing bars	Select the reinforcing bar type.
Anchors to	Select to which part the anchor rods are attached, and how the anchors are attached. Default is **Column** and **Weld**.
Anchors all the same L	Select whether the anchor rods have the same length.
Bottom section	Select the profile from profile catalog.
Nut	Nut profile. Define the height of the nut on the **Anchor rods** tab.
Washer	Washer profile. You can define the direction and the rotation for each washer. Define the thickness of the washer on the **Anchor rods** tab.
Weld washer and nut to anchor	Select whether washers and nuts are welded to the anchors.
Tube top	Tubular embed for creating a round hole for the anchors. The tube top starts at the bottom level of the nut.

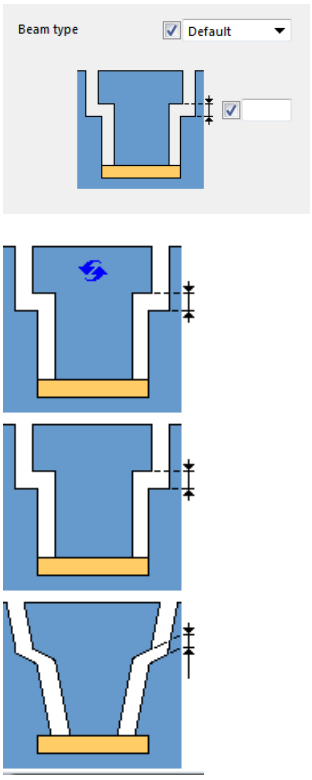
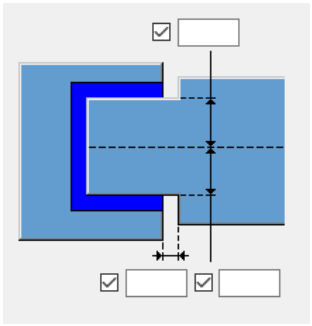
Option	Description
Tube bottom	Tubular embed for creating a round hole for the anchors. The height of the tubular profile can be modified on the **Anchor rods** tab.
Tubes around anchors to beam	Select how the tubes around the anchors are attached to the beam. Default is **Weld**.
Tubes all the same L	Select whether the tubes have the same length.
Tubes aligned with	Select whether the tubes are aligned with the column, top of the beam, or bottom of the beam.
Create cuts around tubes	Select whether cuts are created around the tubes.

Parameters tab

Use the **Parameters** tab to define how the column and the beam are cut (skew/square).

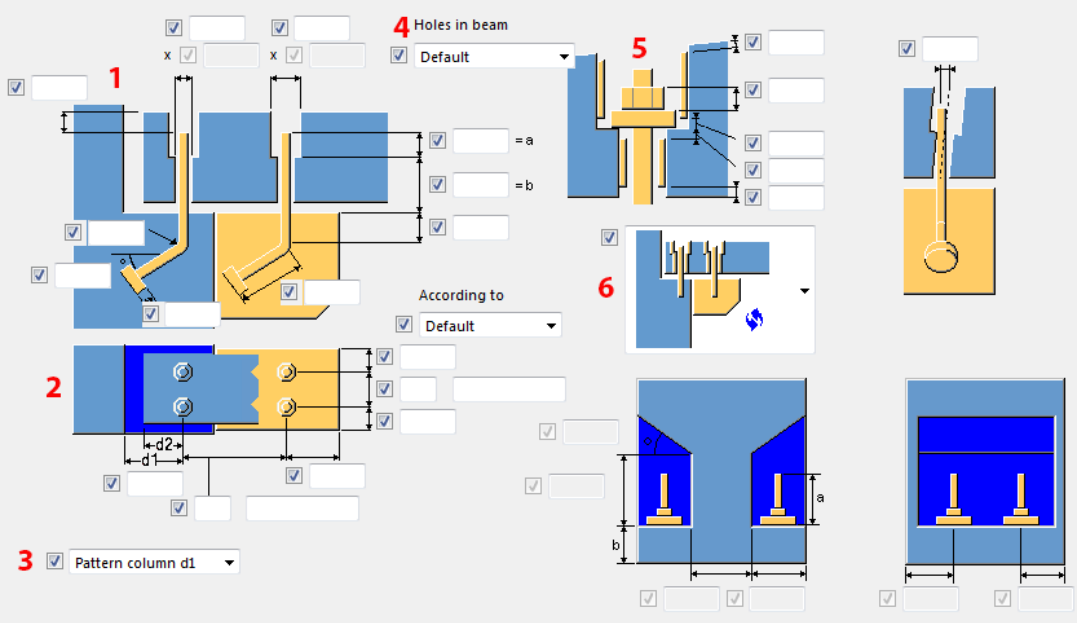
Option	Description	
Fitting to column	Select how the top of the column is fitted. Default is **Perpendicular**.	
Column cut-out		Default. A full-width cut-out is created. 
		Column left side cut-out. Clearance between the

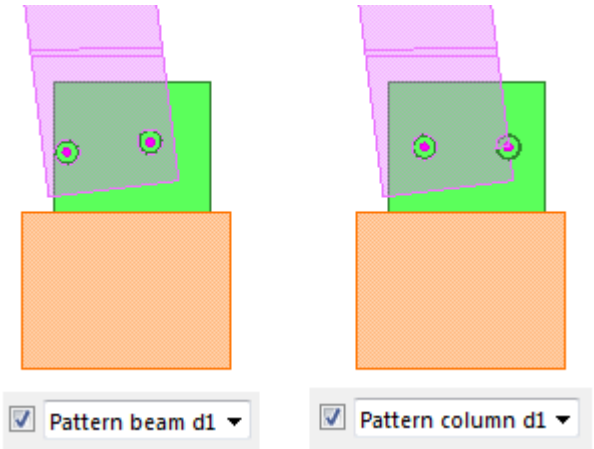
Option	Description	
		column and the beam can be set. 
		Column right side cut-out. Clearance between the column and the beam can be set. 
		Both sides cut-out. Clearance between the column and the beam can be set on both sides. 
Beam end cut-out Use this option when the beam and the column are not aligned.		Square beam end
		Skewed beam end (according to main part)

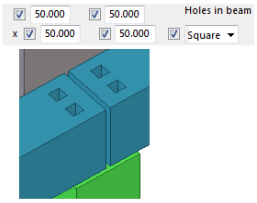
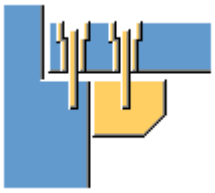
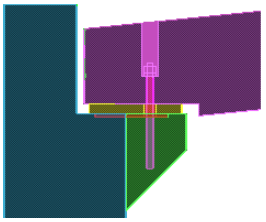
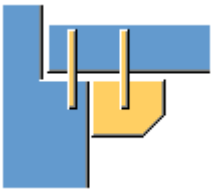
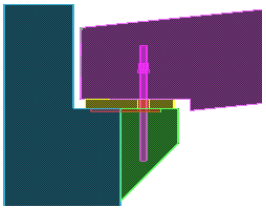
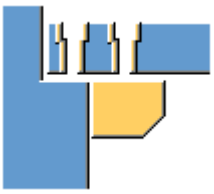
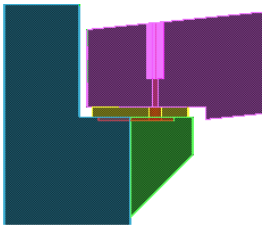
Option	Description	
Beam bottom cut-out Use this option when the beam and the column are not aligned.		Square beam bottom
		Skewed beam bottom (according to main part)
Column cut-out for T-shaped secondary part		In **Beam type**, select the shape of the secondary part. • **Rectangular** - always creates a rectangular column cut-out. • **T-beam** - Tekla Structures checks the shape of the beam and creates a T-shaped column cut-out. • **Automatic** - Tekla Structures automatically checks the shape of the secondary part, and creates either a rectangular or a T-shaped column cut-out. **Automatic** is the default value. For T-shaped cut-outs, define the cut-out dimension between the column and the beam.
Secondary part width		Define the maximum width of the secondary part and cut it to this width.
Cut extra secondary parts	You can find and cut extra secondary parts by using classes or name search.	

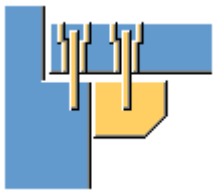
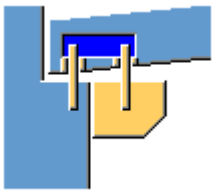
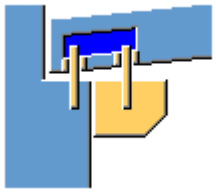
Anchor rods tab

Use the **Anchor rods** tab to define dimensions and position of the anchor rods.



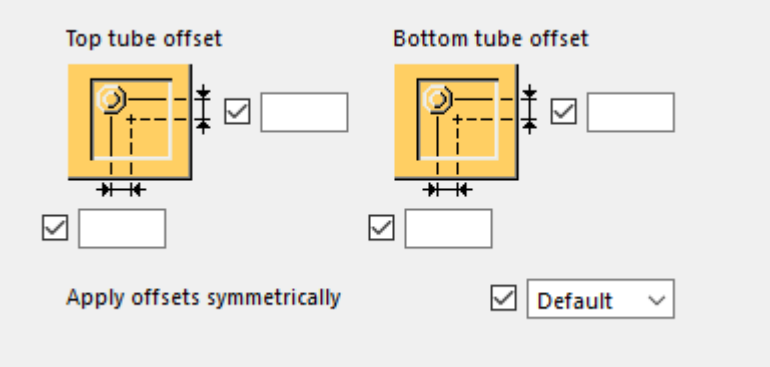
	Description
1	Anchor rod length, hole diameter and top offset.
2	Number of anchor rods, distances and edge distances.
3	Anchor rod distribution options. Useful when the beam is not aligned with the column. 

	Description		
4	Select the type of the hole in the beam. Default is **Circular**. If you set the type to **Circular**, use the two boxes on the left to define the hole diameter. If you set the type to **Square**, use the four boxes on the left to define the dimensions of the square cut.		
5	Height of the tubular profile, nut and washer.		
6	Anchor rods and cuts		Default. Anchor rods are created. Holes for the anchors are created. 
			Anchor rods are created. No holes are created. 
			Only holes are created. No anchors are created. 

	Description		
			Only anchor rods are created. No holes are created.
			Cut-out in the concrete beam. Anchor rods are created. No holes are created.
			Cut-out in the concrete beam (parallel to beam). Anchor rods are created. No holes are created.

Tube offset

Define the offsets for the tube profiles in both the X and Y directions.



Socket tab

Use the **Socket** tab to define whether socket anchors are created, how they are connected and the dimensions and position of the socket anchors.

If you create the sockets on the **Socket** tab, the anchor rods on the **Connection** tab are automatically considered as sockets.

Option	Description
Create socket	Select whether sockets are created and which parts are included.

Option	Description
Connect socket to prim by	Select how the sockets are connected to the main part.
Connection rod-connector	Select the connection method between the rod and the connecting profile.
Type of reinforcing bars	Select the rod type.
Leg rotation	Select the direction of the rod. You can enter an angle in the box on the right.
Legs symmetrically	Select whether custom part sockets are created symmetrically.
Connector **Dowel**	Properties for the connecting profile and the rod.
Partname component	If you want to use a custom part to create the socket, select the **Custom part** option in the **Create socket** list. Then browse for the component, and use the list of options to position the custom part. To use saved custom component properties, select the saved properties file.

General tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Concrete console (111)

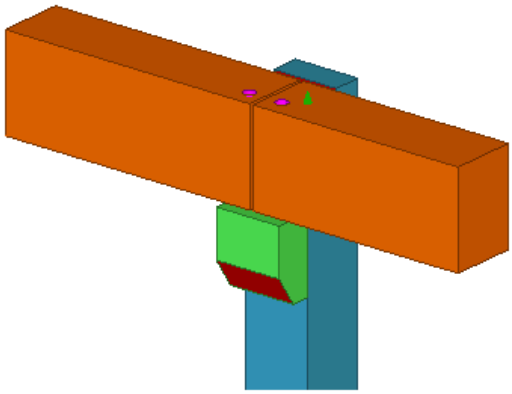
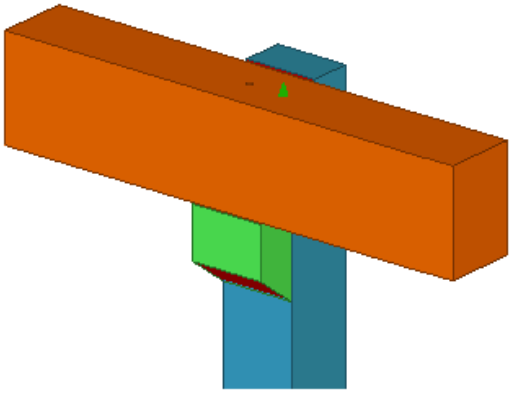
Concrete console (111) creates a connection between a concrete column and two secondary concrete beams. The beams rest on a console which is attached to the column.

Objects created

- Console
- Neoprene strip
- Steel plates

- Anchor rods
- Tubes
- Sockets

Use for

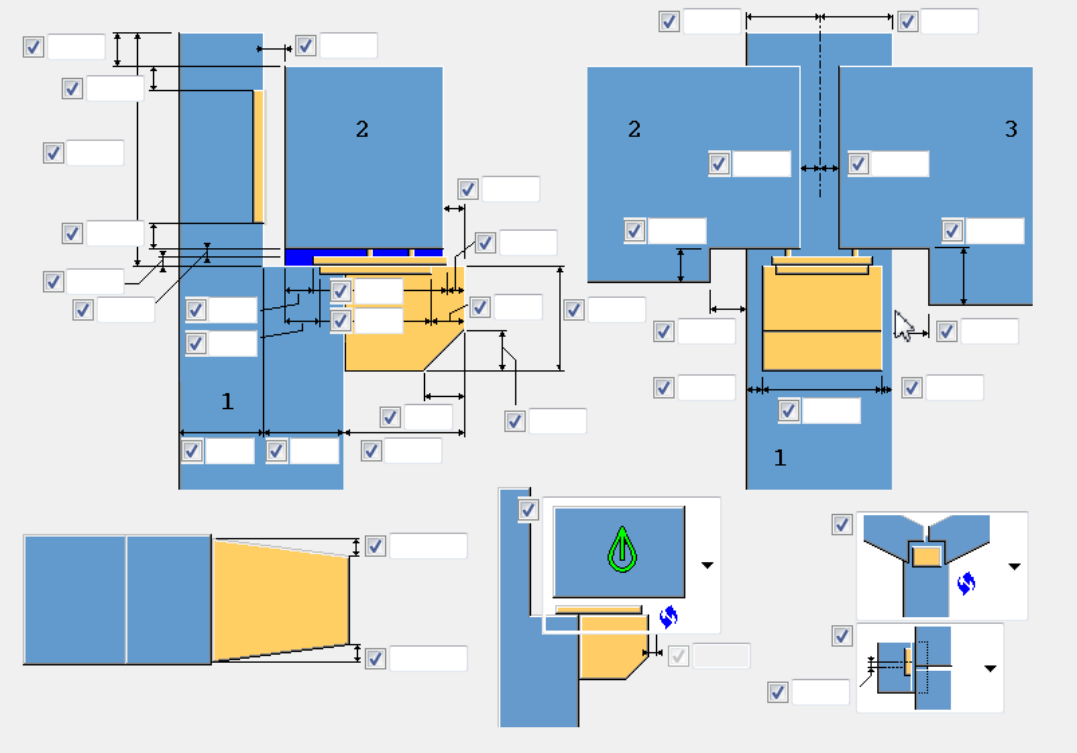
Situation	Description
	Console connection between a concrete column and two concrete beams.
	Console connection between a concrete column and one concrete beam.

Selection order

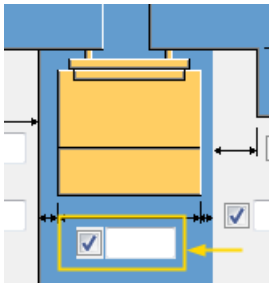
1. Select the main part (column).
2. Select the first secondary part (beam).
3. Select the secondary part (beam).
4. Click the middle mouse button to create the connection.

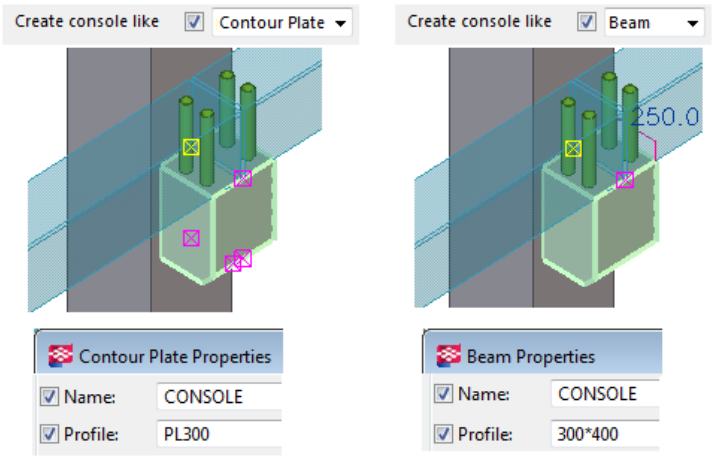
Picture tab

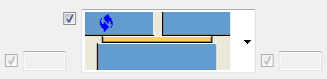
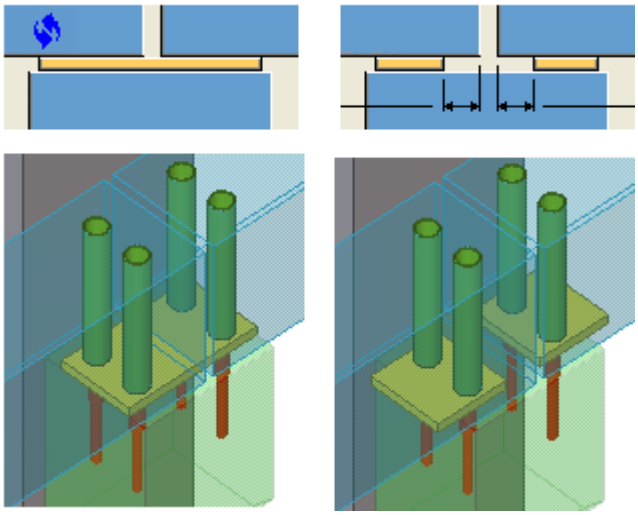
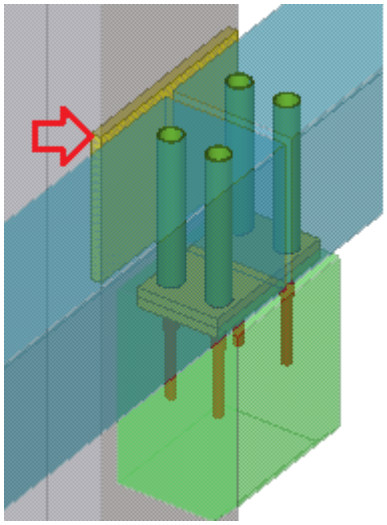
Use the **Picture** tab to define the shape and the dimensions of the console, and the steel support plates and the neoprene layer.

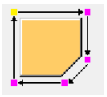


Parts tab
Use the **Parts** tab to define properties for the console, the neoprene layer and the optional steel plates.

Option	Description
Console width	Define the prefix and a start number for the part position number, material, name, class and comment for the console. Define the width on the **Picture** tab.  If no value is entered, the width is the same as the width of the beam.
Console to column	Select how the console is attached to the column.

Option	Description
	Default is **Part Add**. The **No action** option means that the console is a loose part and not attached to any other part in the component.
Create console like	Select the profile type of the console. Default is **Contour plate**. **Contour plate** = the console is created using the **Contour plate** command. **Beam** = the console is created using the **Beam** command. 
Neoprene	Neoprene layer properties. A neoprene plate for shock absorbing and sound-damping can be created between the beam and the console. If a trapezium shaped neoprene block is used, the defined thickness will be the thickness on the column side.
Neoprene to	Select to which part the neoprene layer is attached, and how the neoprene is attached. Default is **Beam** and **Weld**.
Holes in neoprene	Select how the holes in the neoprene part are created. Default is **By bolt**.
Diameter of holes in neoprene	Diameter of the holes in the neoprene part. By default, the hole size in the neoprene part is equal to the holes in the console. Enter a value to overwrite this default hole size.

Option	Description
	Select if the neoprene layer is split for each beam separately. 
Steel plate horizontal	Horizontal steel plate size and properties. The plate is placed under the neoprene part.
Steel plate vertical	Vertical steel plate size and properties. The plate is placed at the console side. 
Add steel plate to the column by	Select how the steel plate is attached to the column. Default is **Weld**.

Option	Description
Rotate console polygon	Define the rotation start point of the console polygon. By default, the rotation starts from point 1. Enter 2, 3, 4, or 5 to change the start point. 
Console polygon direction	Select the direction of the console polygon.

Anchor tab

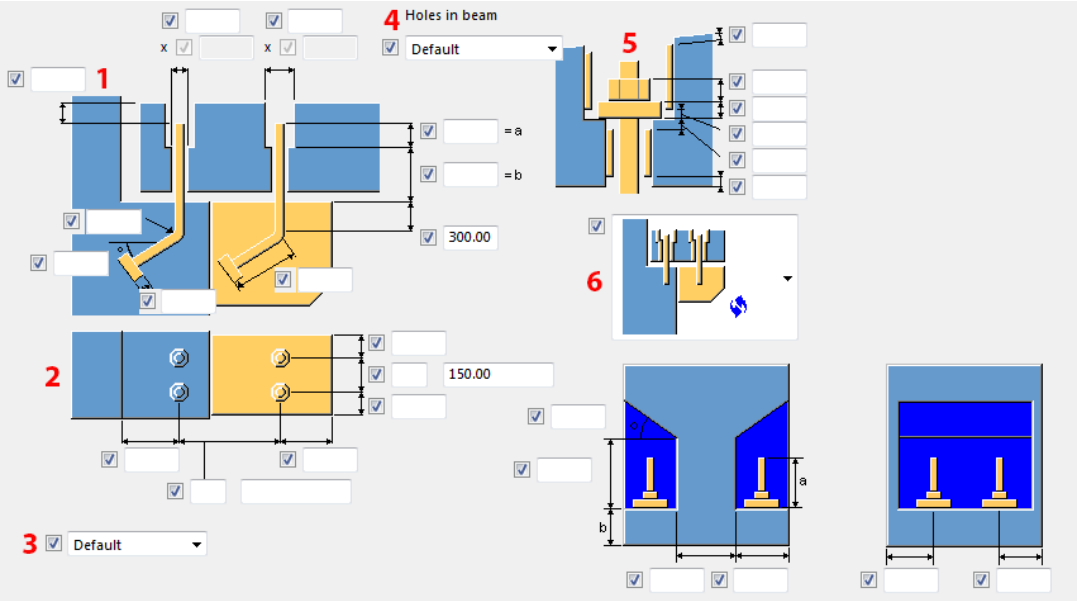
Use the **Anchor** tab to define the properties of the anchor rods, nuts and washers and injection tubes, and select how these parts are connected to the console or the main part.

Option	Description
Anchor rods	Anchor rod profile. Define the length and number of rods on the **Anchor rods** tab.
Type of reinforcing bars	Select the reinforcing bar type.
Anchors to	Select to which part the anchor rods are attached, and how the anchors are attached. Default is **Column** and **Weld**.
Anchors all the same L	Select whether the anchor rods have the same length.
Bottom section	Select the profile from profile catalog.
Washer	Washer profile. Define the thickness of the washer on the **Anchor rods** tab.
Nut	Nut profile. Define the height of the nut on the **Anchor rods** tab.
Weld washer and nut to anchor	Select whether washers and nuts are welded to the anchors.
Tube top	Tubular embed for creating a round hole for the anchors. The tube top starts at the bottom level of the nut.
Tube bottom	Tubular embed for creating a round hole for the anchors. The height of the tubular profile can be modified on the **Anchor rods** tab.

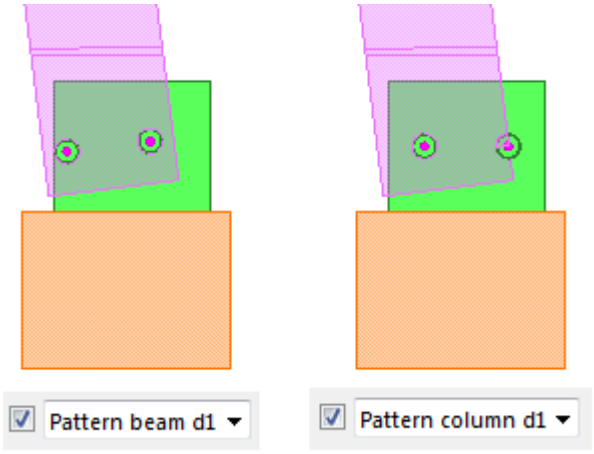
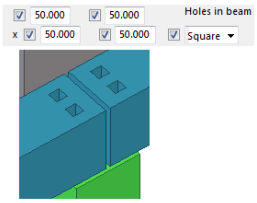
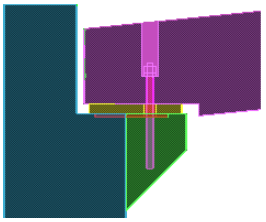
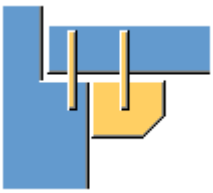
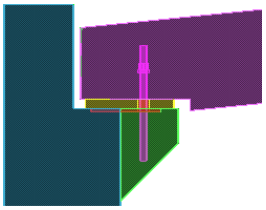
Option	Description
Tubes around anchors to beam	Select how the tubes around the anchors are attached to the beam. Default is **Weld**.
Tubes all the same L	Select whether the tubes have the same length.
Tubes aligned with	Select whether the tubes are aligned with the column, top of the beam, or bottom of the beam.
Create cuts around tubes	Select whether cuts are created around the tubes.

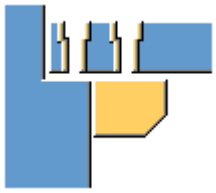
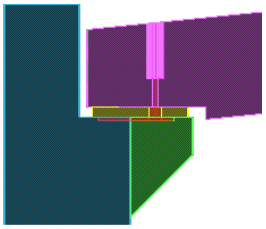
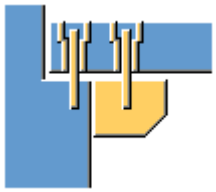
Anchor rods tab

Use the **Anchor rods** tab to define dimensions and position of the anchor rods.



	Description
1	Anchor rod length, hole diameter and top offset.
2	Number of anchor rods, distances and edge distances.
3	Anchor rod distribution options.

	Description		
	Useful when the beam is not aligned with the column. 		
4	Select the type of the hole in the beam. Default is **Circular**. If you set the type to **Circular**, use the two boxes on the left to define the hole diameter. If you set the type to **Square**, use the four boxes on the left to define the dimensions of the square cut.		
5	Height of the tubular profile, nut and washer.		
6	Anchor rods and cuts		Default. Anchor rods are created. Holes for the anchors are created. 
			Anchor rods are created. No holes are created. 

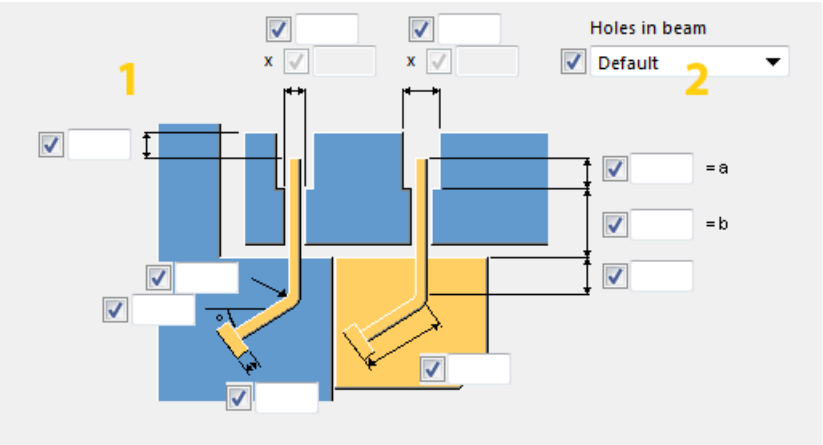
	Description		
			Only holes are created. No anchors are created. 
			Only anchor rods are created. No holes are created.
			Cut-out in the concrete beam. Anchor rods are created. No holes are created.

Anchor rods beam 2 tab

Use the **Anchor rods beam 2** tab to define the properties of the anchor rods, nuts and washers and injection tubes for the second secondary beam. The default profile for the anchor rods created for the second secondary beam is the same as for the anchors created on the **Anchor** tab.

Option	Description
Anchor rods	Anchor rod profile. Define the length and number of rods on the **Anchor rods** tab.
Type of reinforcing bar	Select the reinforcing bar type.
Bottom section	Select the profile from the profile catalog.
Nut	Nut profile. Define the height of the nut on the **Anchor rods** tab.
Washer	Washer profile. Define the thickness of the washer on the **Anchor rods** tab.

Option	Description
Tube top	Tube profile. Tubular embed for creating a round hole for the anchors. The tube top starts at the bottom level of the nut.
Tube bottom	Tube profile. Tubular embed for creating a round hole for the anchors. The height of the tubular profile can be modified on the **Anchor rods** tab.



	Description
1	Anchor rod length, hole diameter and top offset.
2	Select the type of the hole in the beam. If you set the type to **Circular (neg.vol.)**, use the two boxes on the left to define the hole diameter. **Circular (neg.vol.)** is the default value. If you set the type to **Circular (drilling)**, use the two boxes on the left to define the hole diameter. If you set the type to **Square**, use the four boxes on the left to define the dimensions of the square cut.

Socket tab
Use the **Socket** tab to define whether socket anchors are created, how they are connected and the dimensions and position of the socket anchors.

If you create the sockets on the **Socket** tab, the anchor rods on the **Anchor** tab are automatically considered as sockets.

Option	Description
Create socket	Select whether sockets are created and which parts are included.
Connect socket to prim by	Select how the sockets are connected to the main part.
Connection rod-connector	Select the connection method between the rod and the connecting profile.
Type of reinforcing bars	Select the rod type.
Leg rotation	Select the direction of the sockets. You can enter an angle in the box on the right.
Connecting profile **Rod**	Properties for the connecting profile and the rod.
Partname component	If you want to use a custom part to create the socket, select the **Custom part** option in the **Create socket** list. Then browse for the component, and use the list of options to position the custom part. To use saved custom component properties, select the saved properties file.

General tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Concrete beam-beam (112)

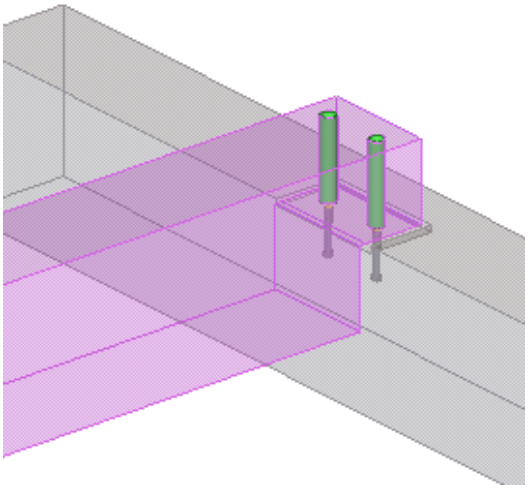
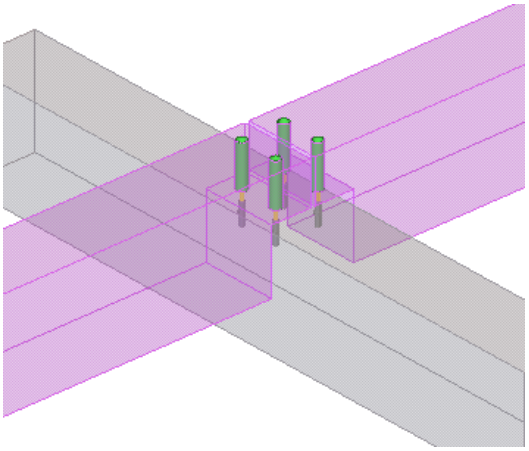
Concrete beam - beam (112) creates a connection between a concrete beam and one or two secondary concrete beams.

Objects created

- Neoprene
- Steel plates

- Anchor rods
- Tubes
- Sockets

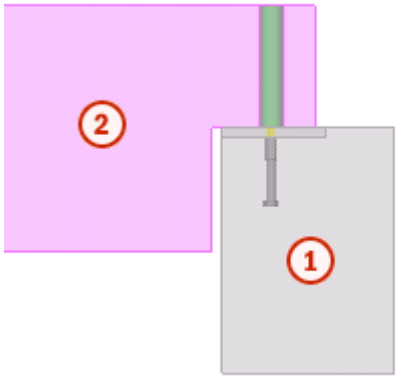
Use for

Situation	Description
	Connection between two concrete beams.
	Connection between three concrete beams.

Selection order

1. Select the main part (beam).
2. Select one or two secondary parts (beam).
3. Click the middle mouse button to create the connection.

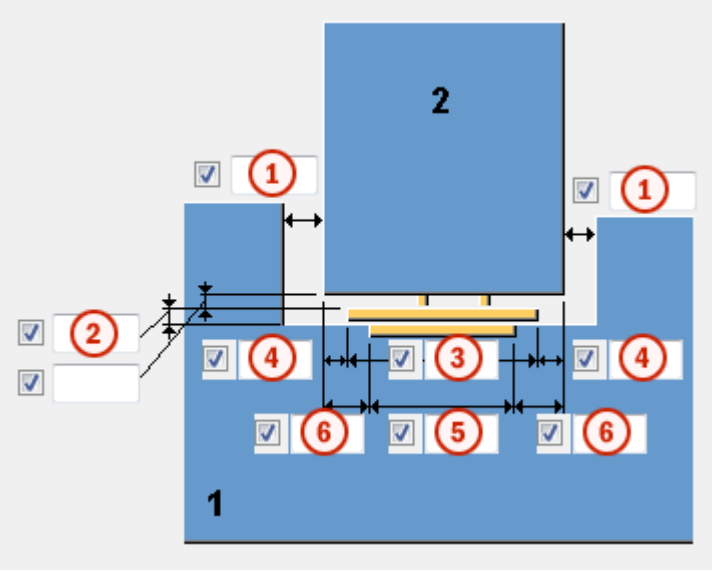
Part identification key

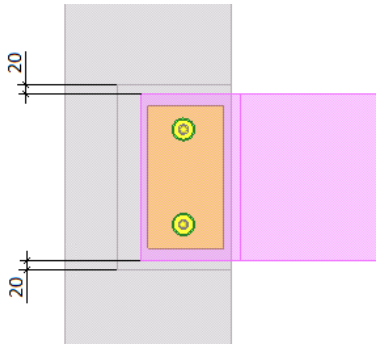
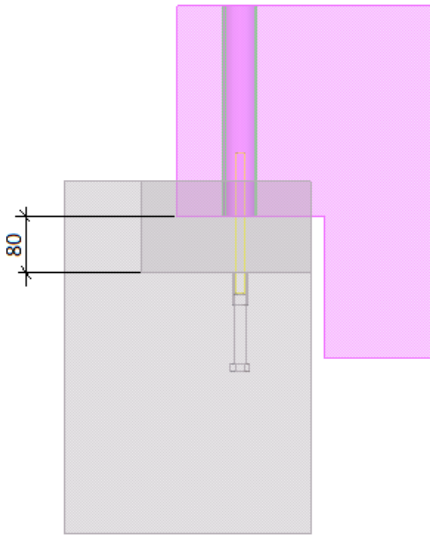
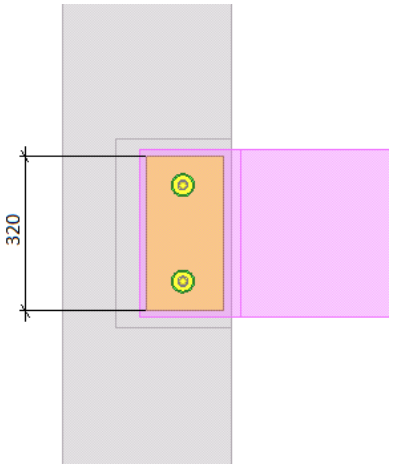


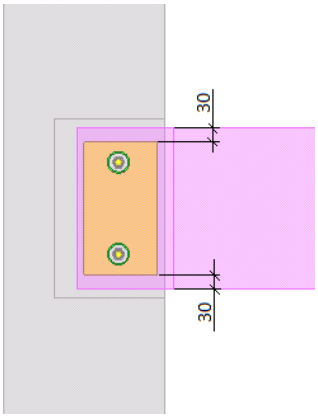
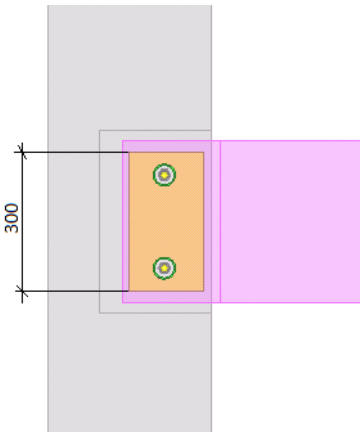
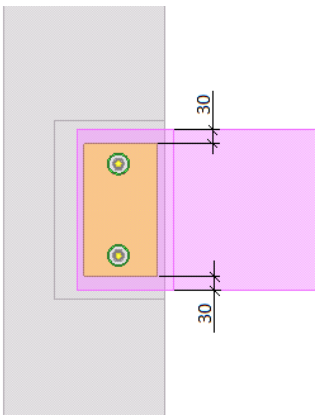
	Part
1	Beam
2	Beam

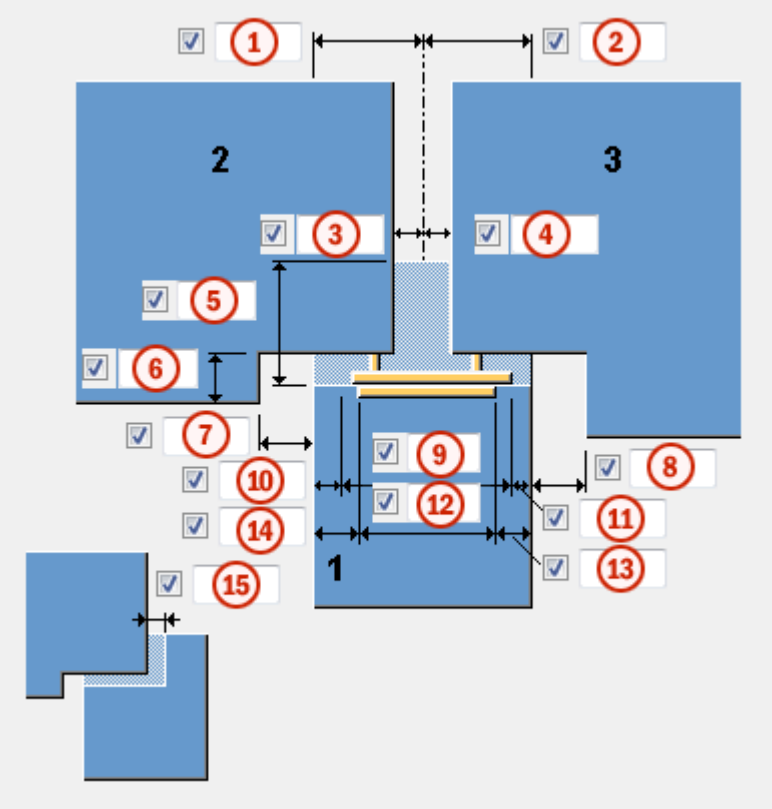
Picture tab
Use the **Picture** tab to control the part dimensions and the shapes of the parts, and the recesses.

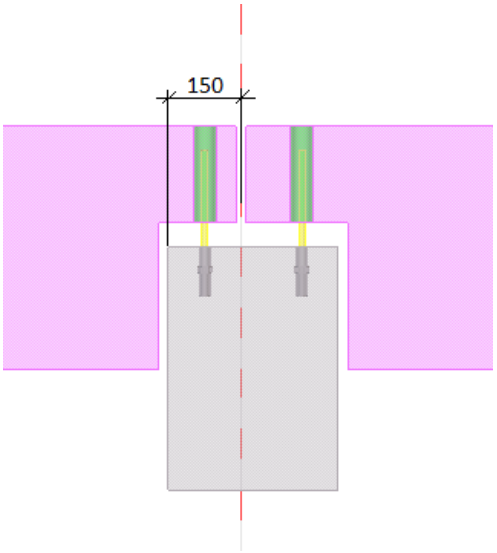
Dimensions

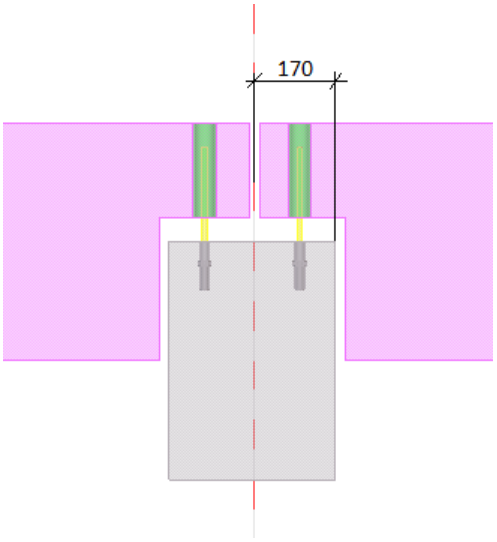
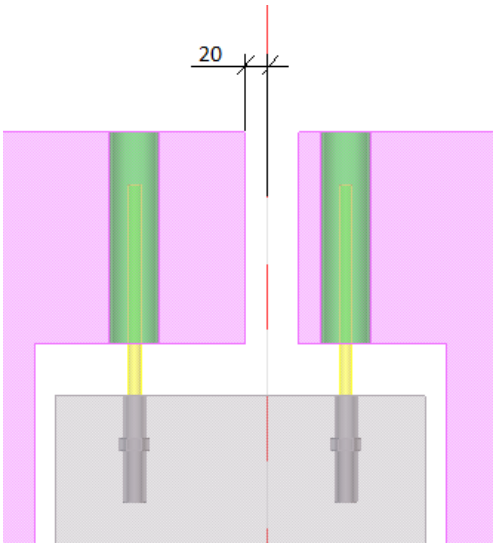


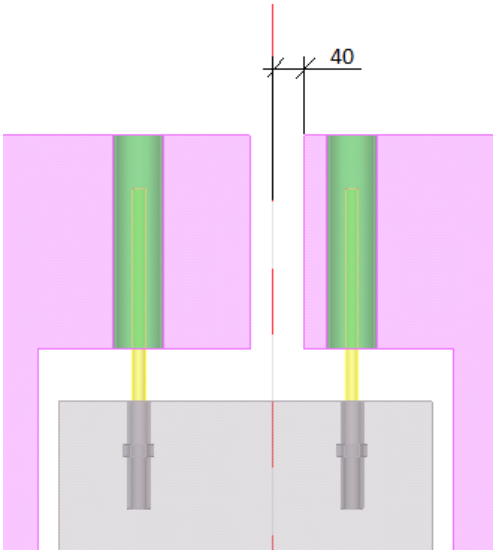
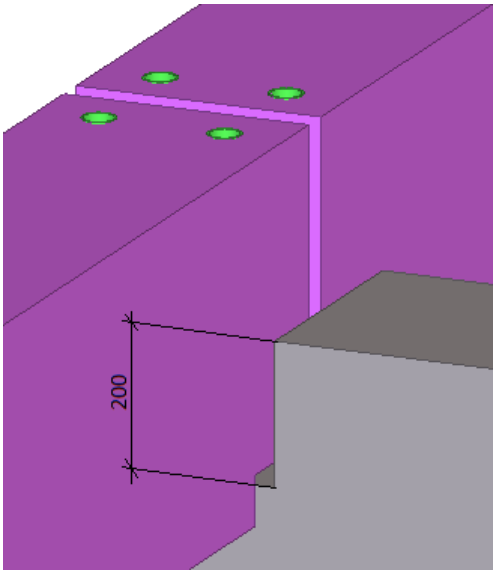
	Description	Example
1	Define the gap between the main part and the secondary part on the left and the right side.	
2	Define the vertical gap between the main part and the secondary part.	
3	Define the length of the neoprene.	

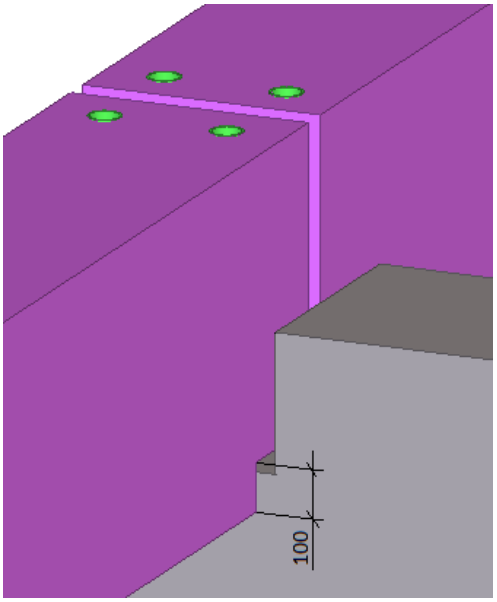
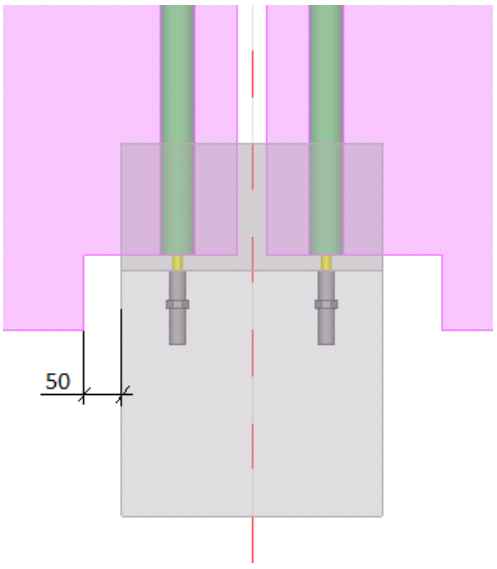
	Description	Example
4	Define the gap between the neoprene and the secondary part on the left and the right side.	
5	Define the length of the steel plate.	
6	Define the gap between the steel plate and the secondary part on the left and the right side.	

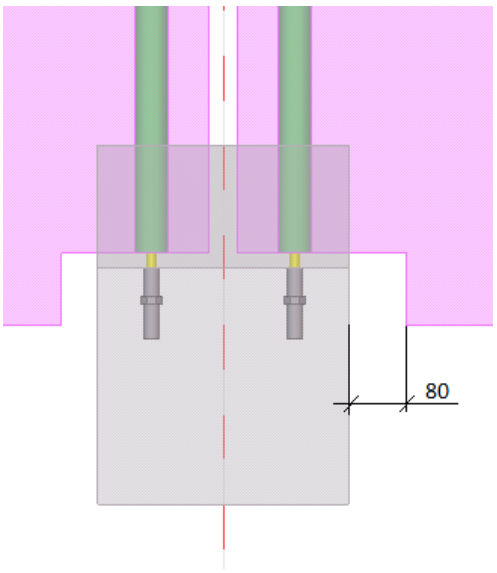
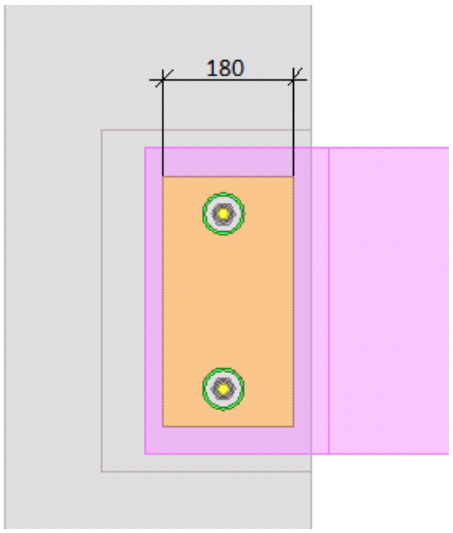
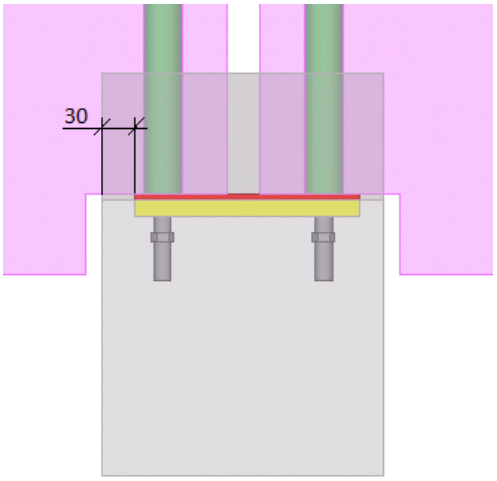


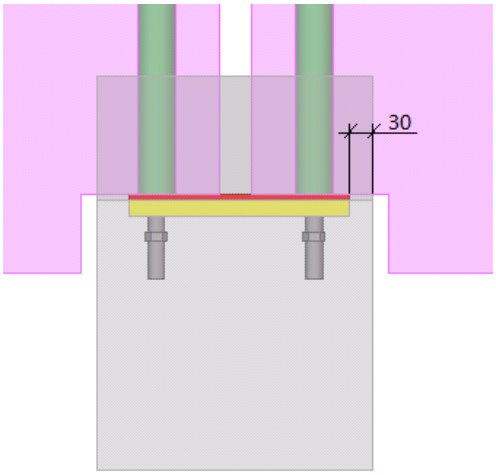
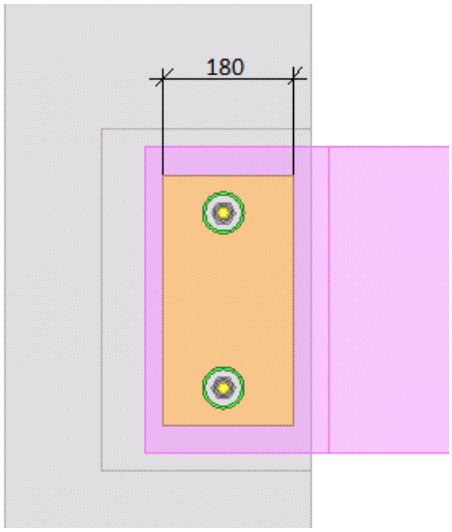
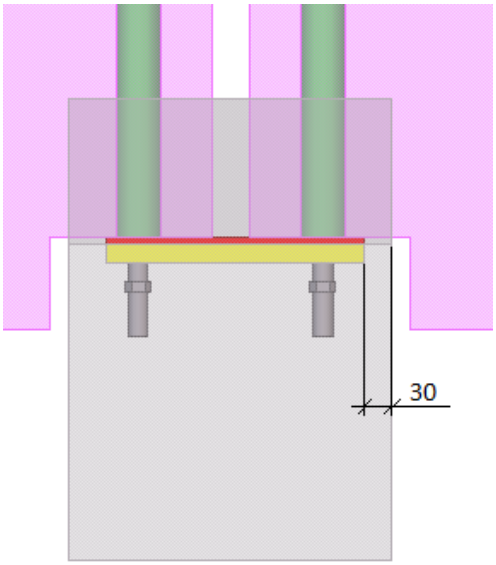
	Description	Example
1	Define the distance from the centerline of the secondary parts to the outer contour of the main part on the left side.	

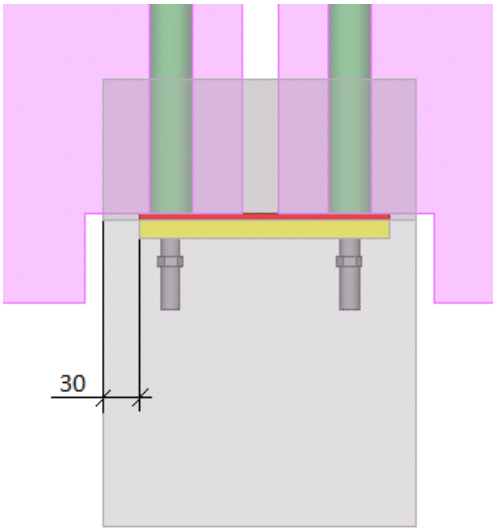
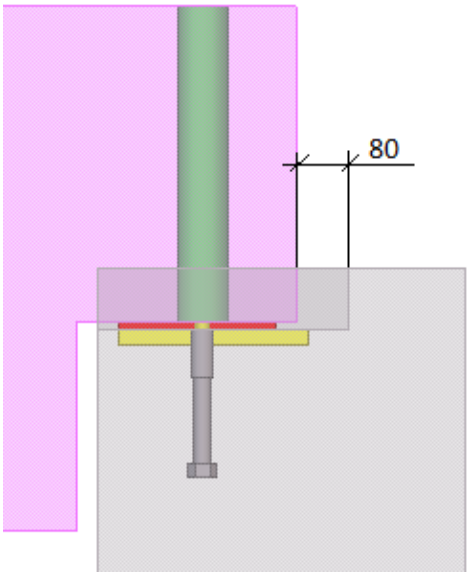
	Description	Example
2	Define the distance from the centerline of the secondary parts to the outer contour of the main part on the right side.	
3	Define the distance from the edge of the first secondary part to the centerline.	

	Description	Example
4	Define the distance from the edge of the second secondary part to the centerline.	
5	Define the depth of the recess in the main part.	

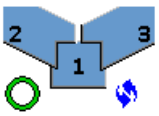
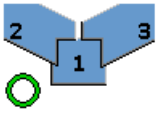
	Description	Example
6	Define the depth of the recess in the secondary part.	
7	Define the width of the recess in the first secondary part.	

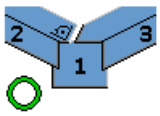
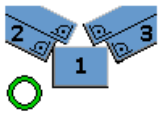
	Description	Example
8	Define the width of the recess in the second secondary part.	
9	Define the width of the neoprene.	
10	Define the distance from the edge of the neoprene to the edge of the main part.	

	Description	Example
11	Define the distance from the edge of the neoprene to the edge of the main part.	
12	Define the width of the steel plate.	
13	Define the distance from the edge of the steel plate to the edge of the main part.	

	Description	Example
14	Define the distance from the edge of the neoprene to the edge of the main part.	
15	Define the distance from the edge of the recess in the main part to the edge of the secondary part.	

Fit secondaries perpendicular to

Option	Description
	Default Secondary parts are fitted perpendicular to the main part. AutoDefaults can change this option.
	Secondary parts are fitted perpendicular to the main part.

Option	Description
	The top side of secondary parts is fitted perpendicular to secondary parts while the bottom side of secondary parts is fitted perpendicular to the main part.
	Both the top side and the bottom side secondary parts are fitted perpendicular to secondary parts.

Parts tab

Use the **Parts** tab to control the neoprene and steel plate properties and how these parts are connected.

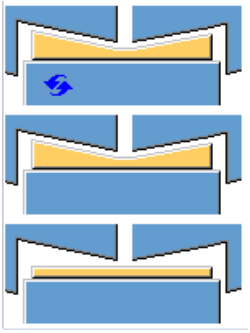
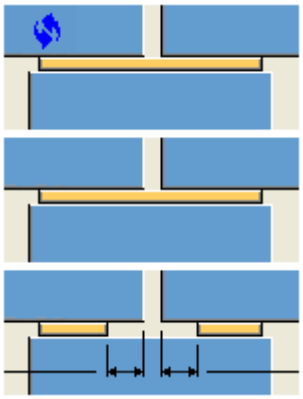
Neoprene and steel plate

Option	Description
Neoprene	Define the neoprene thickness, width and height.
Steel plate	Define the steel plate thickness, width and height.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Comment	Add a comment about the part.	

Neoprene properties

A neoprene plate for absorbing shock and for damping sound can be used between the main and the secondary parts.

Option	Description
Neoprene to	Define to which part the neoprene is connected. The neoprene can also be a loose part. In the second list, define how the neoprene is connected to the part.
Holes in Neoprene	Define how the holes in the neoprene are created.
Diameter of Holes in Neoprene	Define the diameter of the holes in the neoprene.
	Define whether the neoprene is fitted to fill the cut space between the main part and the secondary part, or whether it is created in a rectangular shape.
	Define whether the neoprene is created as one single plate or as two plates, and the edge distance from the neoprene to the secondary parts.

Steel plate properties

One or two additional steel plates can be used between the main and the secondary parts.

Option	Description
Add steel plate to the primary by	Define how the steel plate is connected to the main part.
Negative volume around steel plate	Define whether there is a negative volume around the steel plate.

Anchors tab
Use the **Anchors** tab to control the properties of anchor rods, nuts, washers and the top and bottom tubes.

Option	Description
Anchor rods	Anchor rods can be used between the main and the secondary parts. Select the anchor rod profile from the profile catalog. If you set the **Type of reinforcing bars** option to **Reinforcing bar**, select the reinforcing bar from the catalog.
Nut	Select the nut profile from the profile catalog.
Washer	Select the washer profile from the profile catalog.
Tube top, **Tube bottom**	Tubes can be used between the main and the secondary parts. Select the profile from the profile catalog.

Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Comment	Add a comment about the part.	

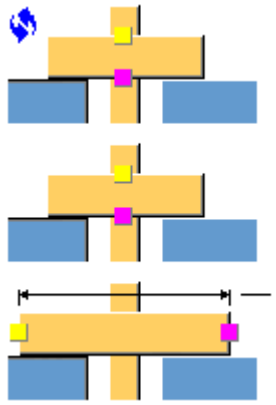
Anchor rods

Type of reinforcing bars	Define the type of reinforcing bars.

Anchors to	Define to which part the anchor rods are connected. The anchor rods can also be loose parts. In the second list, define how the anchor rods are connected to the part.
Anchors all the same L	Define whether the anchor rods have the same length.
Bottom section	Define the type of profile in the bottom section.
Anchors only in secondary	Select whether anchor rods are included only in the secondary parts.

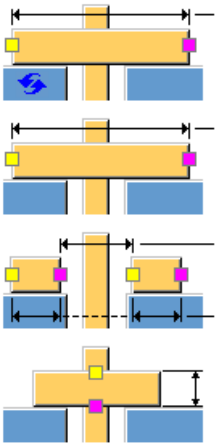
Washer

Define the position of the washer handles.



Washer plate

Define the position of the plate handles.



Option	Description
Create washers symmetrically	Define whether the washers on the anchors are created symmetrically (mirrored) to the washers on the anchors at the opposite beam side.
Weld washer and nut to anchor	Define whether the washers and nuts are welded to the plates.

Tube top and tube bottom

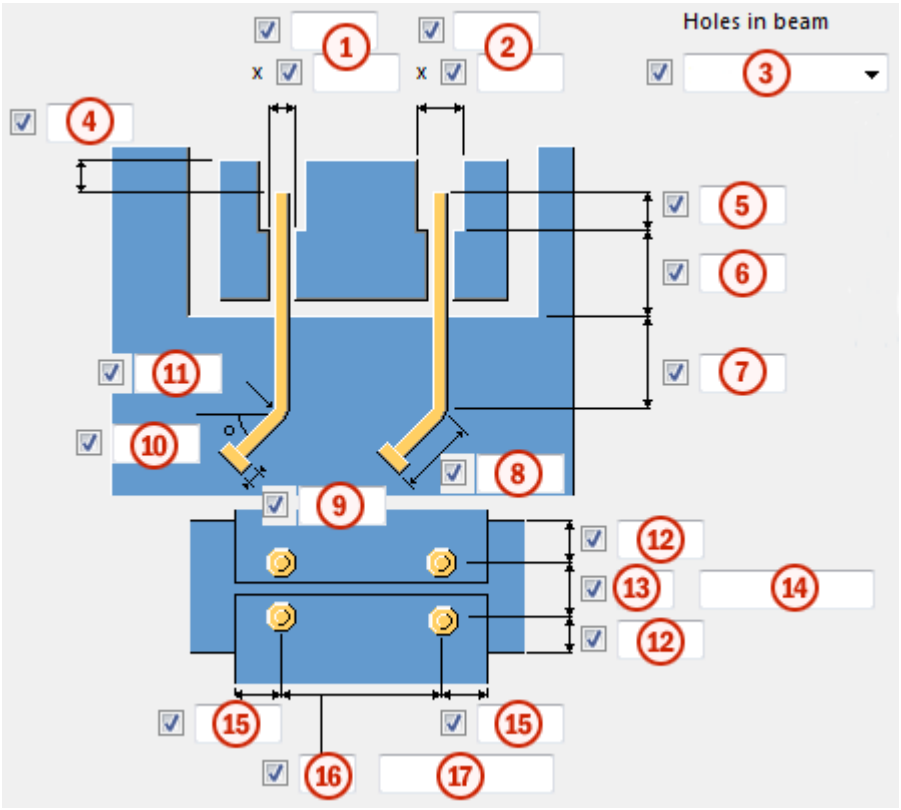
You can add a tubular embed for creating a round hole for the anchors. The tube top starts at the bottom level of the nut.

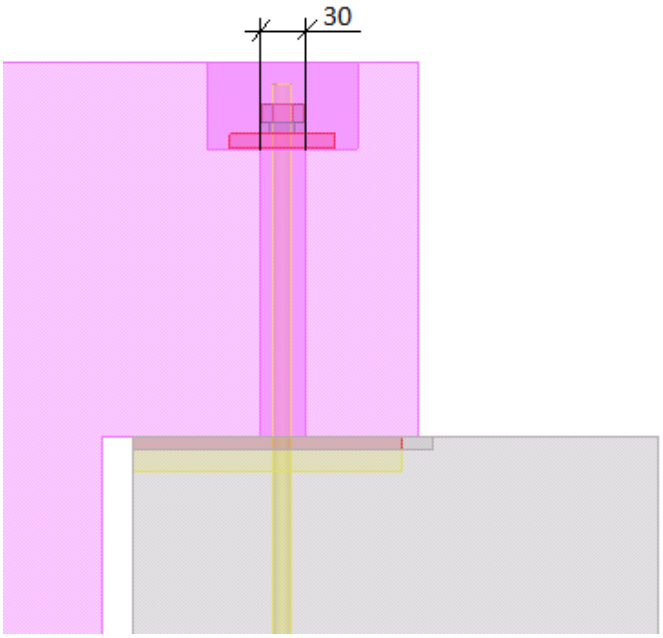
Option	Description
Tubes around anchors to beam	Define how to connect the tubes to the beam.
Tubes all the same L	Define whether the tubes have the same length.
Tubes aligned with	Select whether the tubes are aligned with the main part, or with the top or bottom of the secondary part.
Create cuts around tubes	Select whether cuts are created around the tubes.

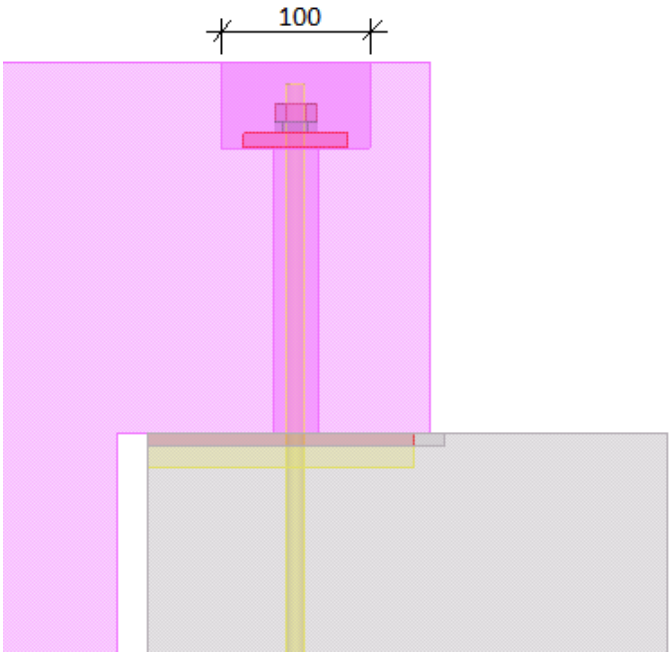
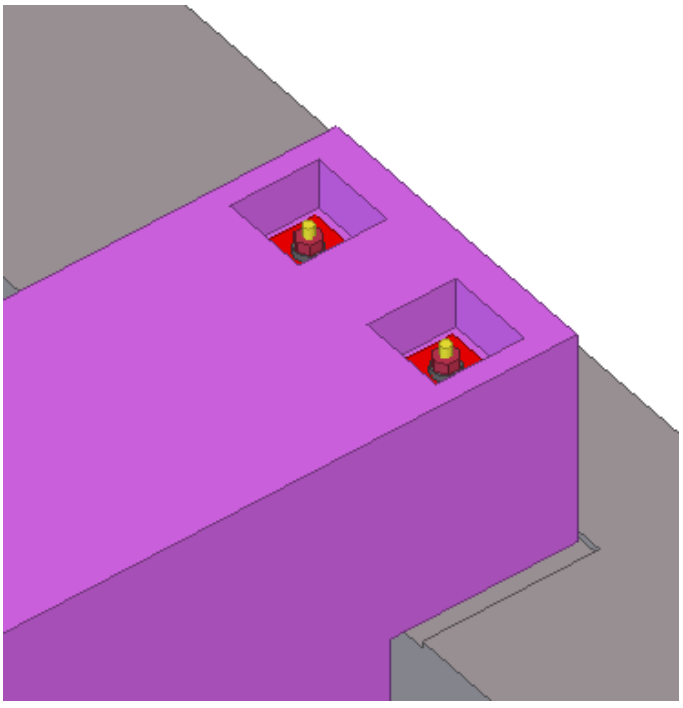
Anchor rods tab

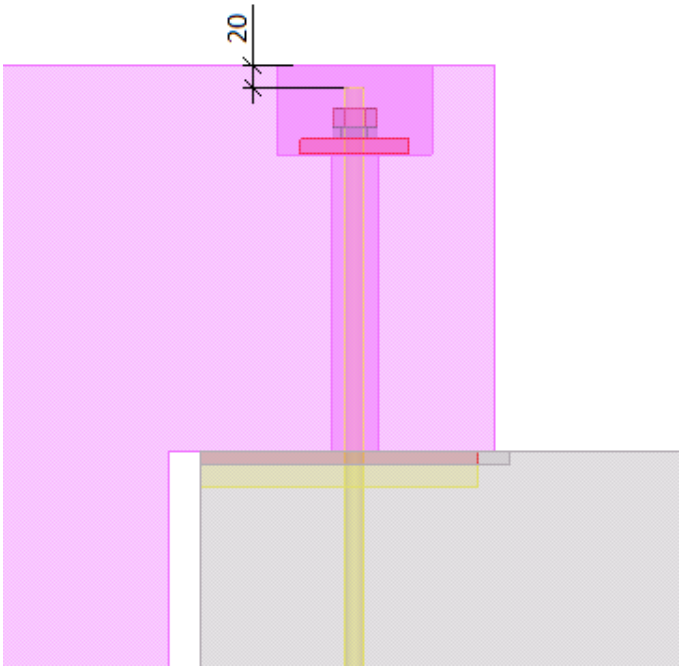
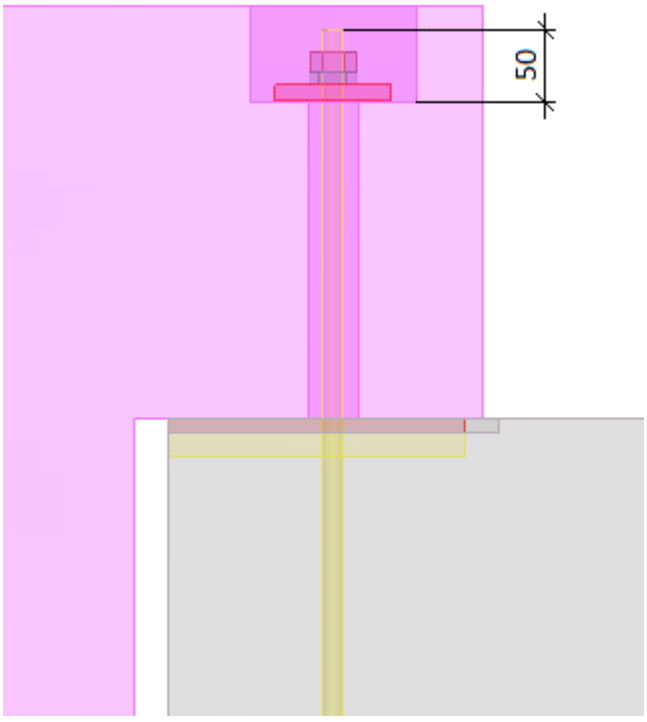
Use the **Anchor rods** tab to control the number, dimensions and position of the anchor rods and the dimensions of the recesses in the main and secondary parts.

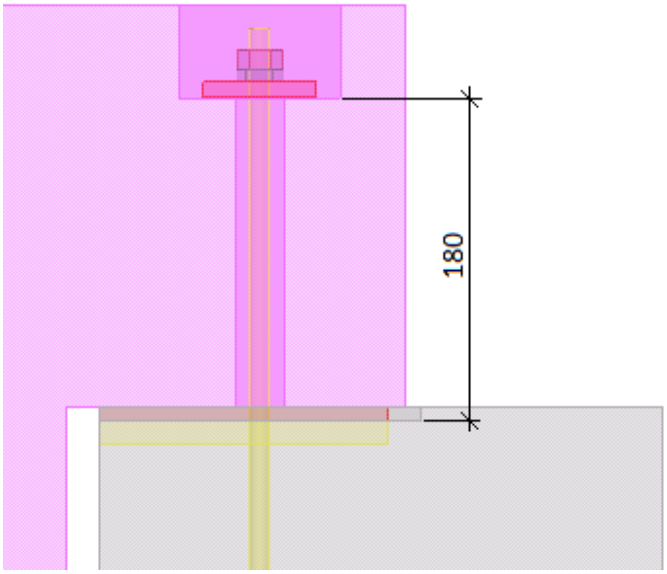
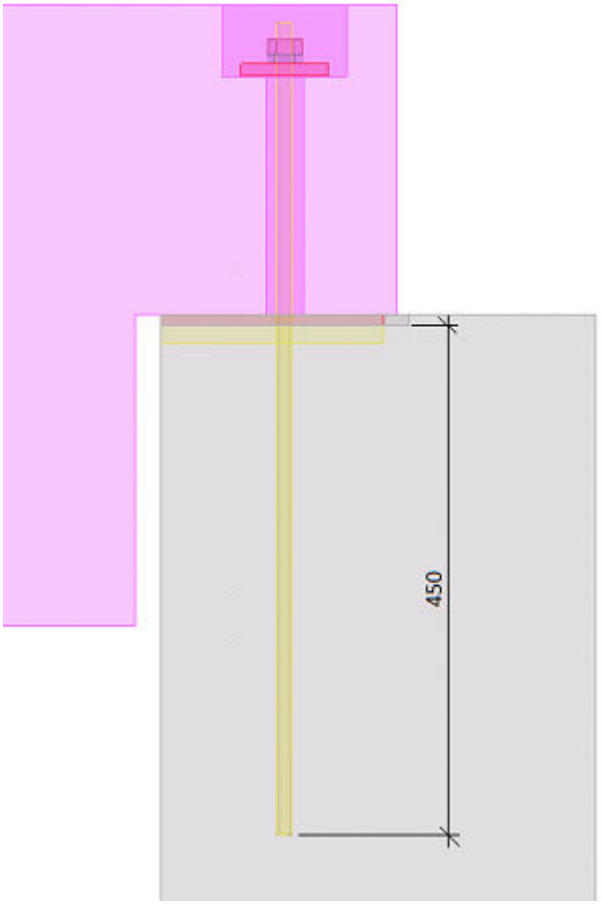
Dimensions

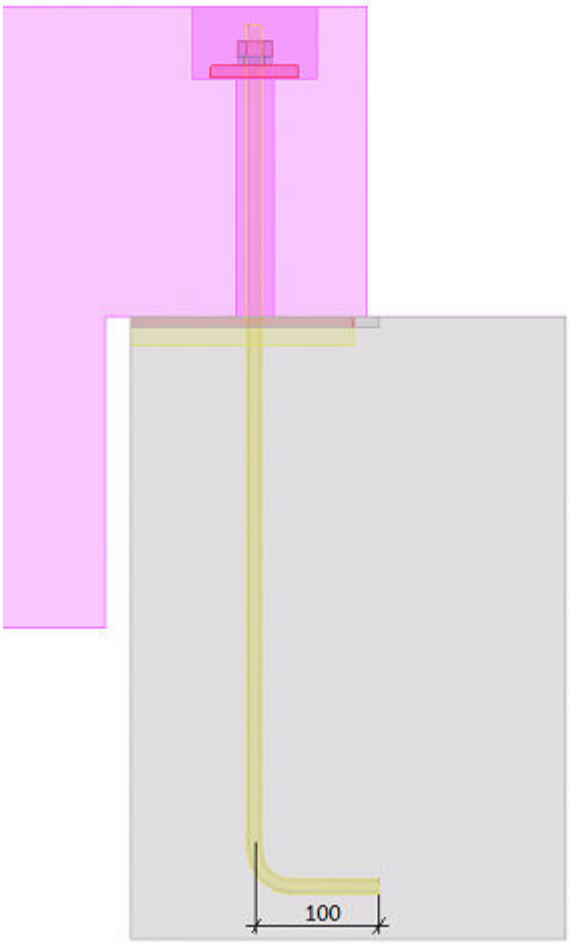


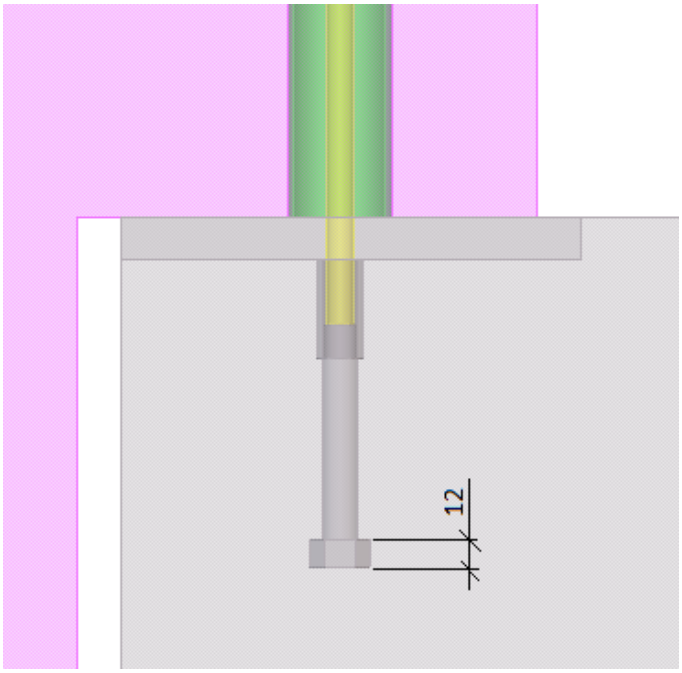
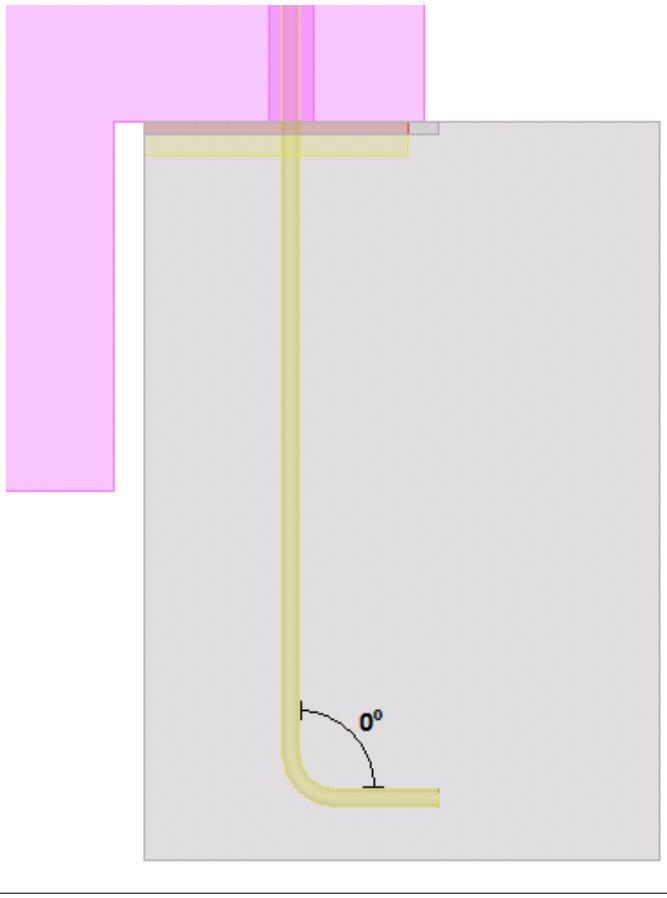
	Description	Example
1	Define the size of the holes in the beam.	

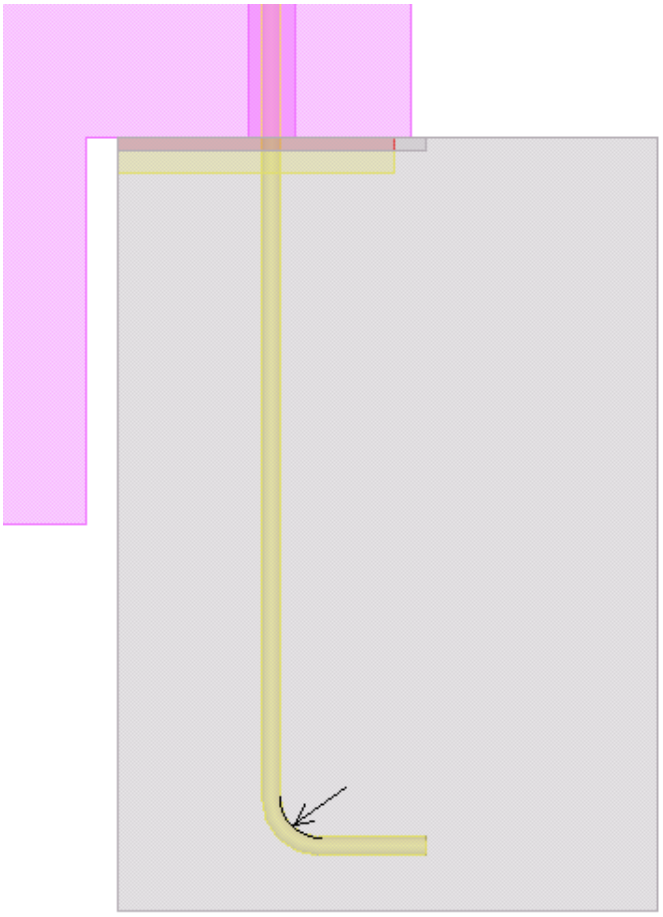
	Description	Example
2	Define the size of the recesses in the beam in both directions.	
3	Define the type of the holes in the beam.	

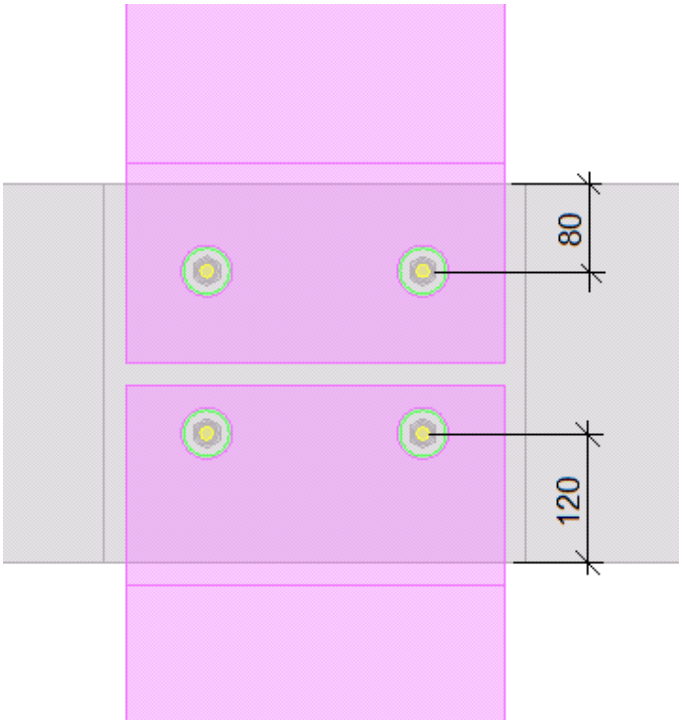
	Description	Example
4	Define the distance between the top of the anchor rod and the top of the beam.	
5	Define the distance between the bottom of the recess and the top of the anchor rod.	

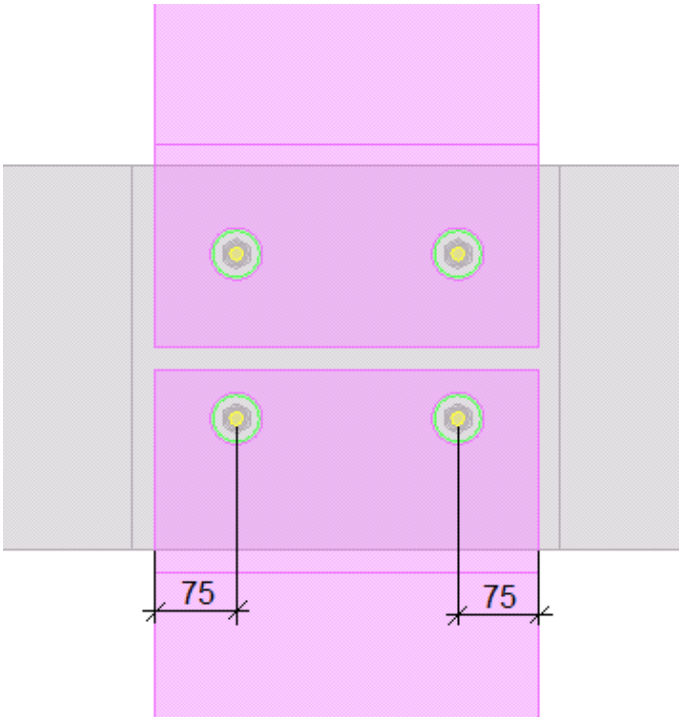
	Description	Example
6	Define the distance between the bottom of the recess in the main part and the bottom of the recess in the secondary part.	
7	Define the distance from the centerline of the secondary parts to the outer contour of the main part.	

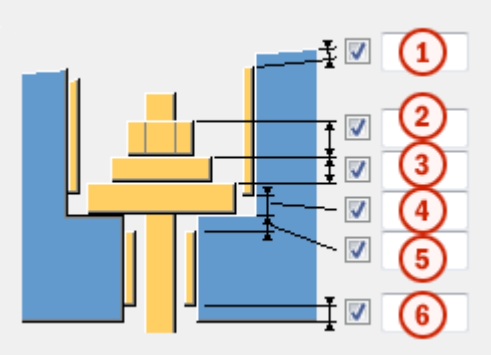
	Description	Example
8	Define the width of the anchor rod hook.	

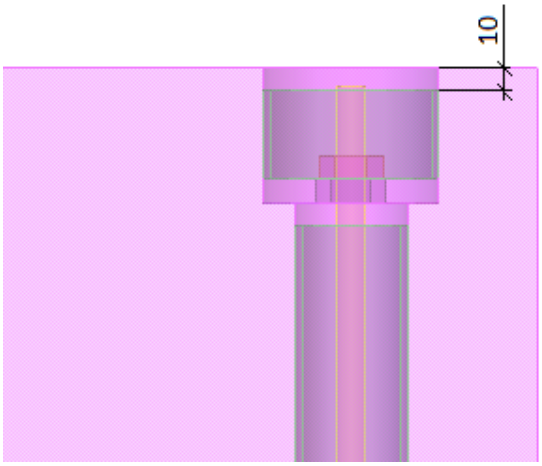
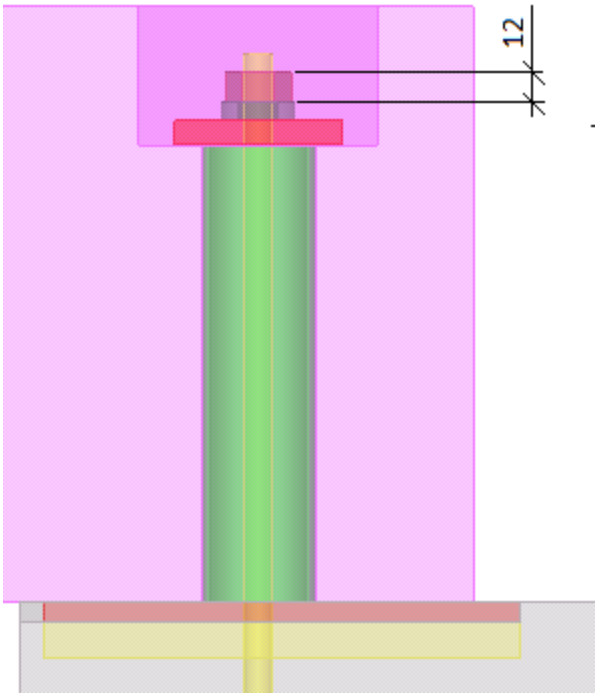
	Description	Example
9	Define the height of the anchor.	
10	Define the angle of the anchor rod hook.	

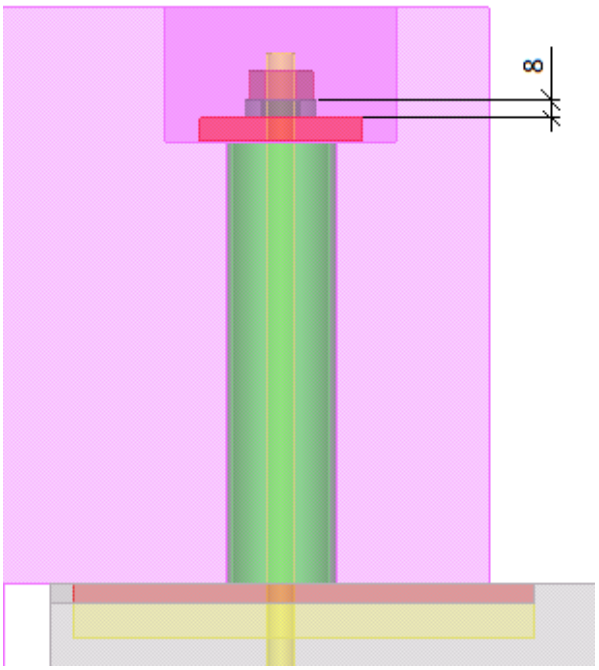
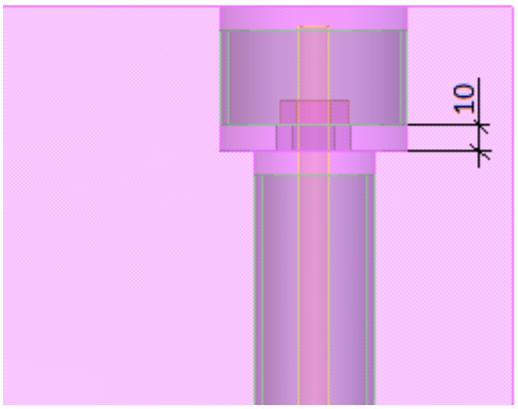
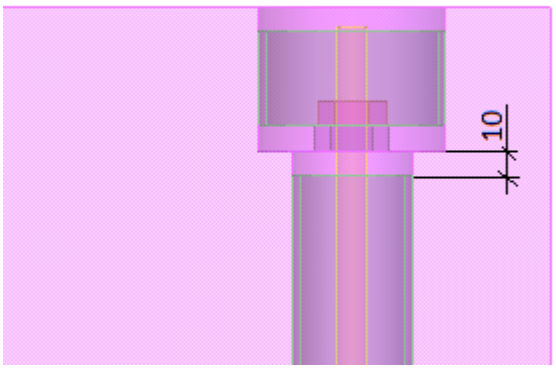
	Description	Example
11	Define the radius of the anchor rod hook in the main part in the vertical direction.	

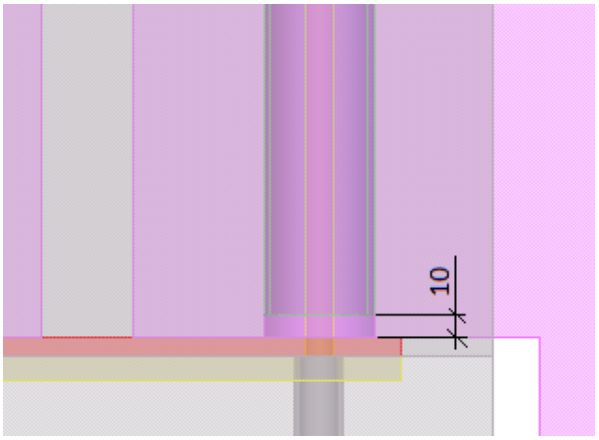
	Description	Example
12	Define the anchor rod edge distances from the main part.	
13	Define the number of anchor rods.	
14	Define the anchor rod spacing. Use a space to separate anchor rod spacing values. Enter a value for each space between anchor rods. For example, if there are 3 anchor rod rows, enter 2 values.	

	Description	Example
15	Define the anchor rod edge distances from the secondary parts.	
16	Define the number of anchors rods.	
17	Define the anchor rod spacing. Use a space to separate anchor rod spacing values. Enter a value for each space between anchor rods. For example, if there are 3 anchor rod rows, enter 2 values.	



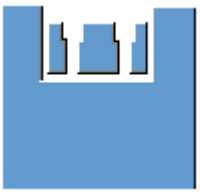
	Description	Example
1	Define the distance between the top side of the top tube and the top of the part.	
2	Define the height of the nut.	

	Description	Example
3	Define the height of the washer.	
4	Define the distance between the bottom side of the top tube and the bottom side of the recess.	
5	Define the distance between the top side of the bottom tube and the bottom side of the recess.	

	Description	Example
6	Define the distance between the bottom side of the bottom tube and the bottom side of the part.	

Recesses

Define whether anchors and recesses are created.

Option	Description
	Anchors and recesses
	Anchors
	Recesses

In the list below the options, select whether the position of anchor rods is calculated in the coordinate system of the main part or the secondary part.

Socket tab

Use the **Socket** tab to control the socket properties and how the sockets are connected. If you create the sockets on the **Socket** tab, the anchors on the **Anchors rods** tab are automatically considered as sockets.

Sockets

Option	Description
Create socket	Select whether sockets are created and which parts are included.
Connect socket to prim by	Select how the sockets are connected to the main part.
Connection rod-connector	Select how to connect the rod connector.
Type of reinforcing bars	Select the rod type.
Leg rotation	Select the direction of the sockets. In the second list, define the angle of the sockets.

Connecting profile, Rod

Option	Description
Connecting profile	Select the profile from the profile catalog.
Rod	Select the rod profile from the profile catalog. If you set the **Type of reinforcing bars** option to **Reinforcing bar**, select the reinforcing bar from the catalog.

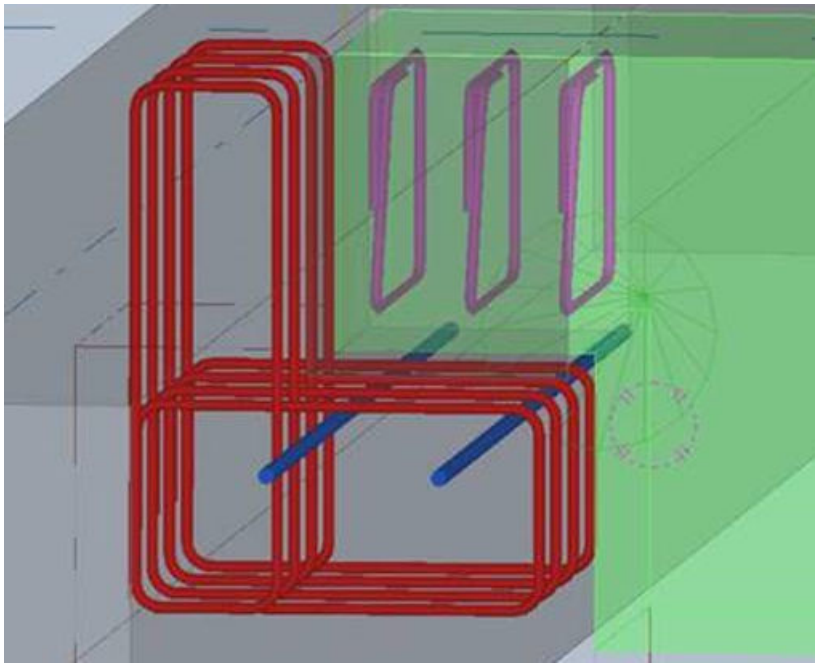
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options**.
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Comment	Add a comment about the part.	

Partname component

Option	Description
Partname component	If you use a custom component to create the socket, browse for the component and use the list of options to position the custom component. To use saved custom component properties, select the saved properties file.

Rebars in primary tab / Rebars in secondary tab

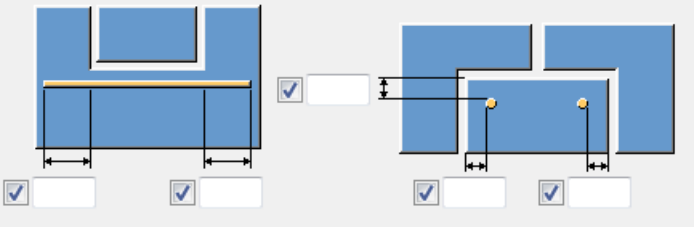
Use the **Rebars in primary** tab and **Rebars in secondary** tab to create reinforcing bars and stirrups at the connection position. On the **Rebars in primary** tab you can create longitudinal reinforcing bars and stirrups for the main part, and on the **Rebars in secondary** tab you can define stirrups for the secondary part.

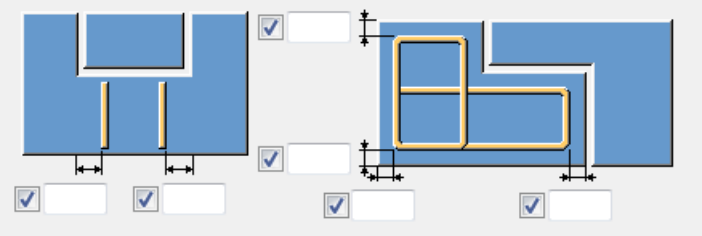


Rebars in primary

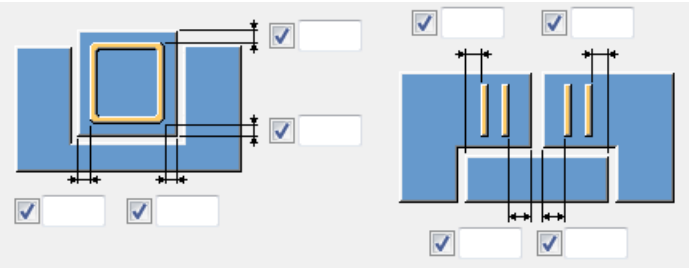
Create the rebars for the main part either by defining the number of rebars, or by setting the spacing or the exact space.

Set the rebar properties, and define the cover thickness and leg length from the edge of the opening.





Rebars in secondary



General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Panels and walls

This section introduces components that can be used in concrete panels and walls.

Click the links below to find out more:

- Wall to wall connection (page 3439)
- Wall groove seam detail (page 3445)
- Anchor (10) (page 3451)

- Wall wall teeth (12) (page 3465)
- Electric box in wall (84) (page 3473)
- Sandwich and double wall (page 3488)
- Sandwich wall horizontal seam (page 3516)
- Sandwich wall vertical seam (page 3524)
- Sandwich wall window (page 3531)
- Wall layout tools (page 3553)
- Geometry detailing strip (page 3584)

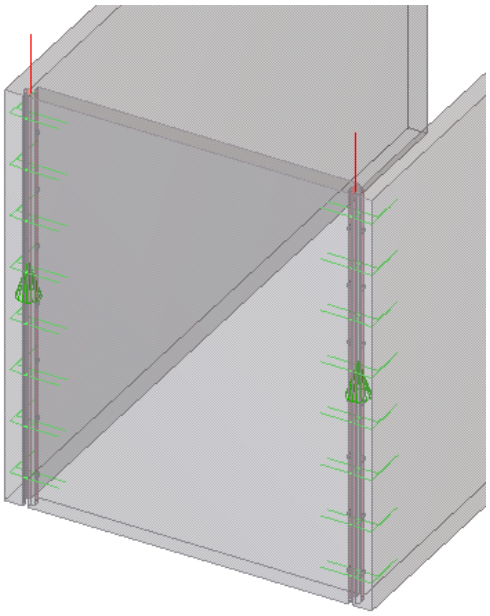
Wall to wall connection

Wall to wall connection creates a connection between two precast walls. The connection can contain a groove with or without additional shear teeth. Additionally, the connection can create reinforcing bar loops or embeds that work as connectors in the walls.

Objects created

- Groove
- Shear teeth or shear cups
- U bars
- Embeds
- Long bar

Use for

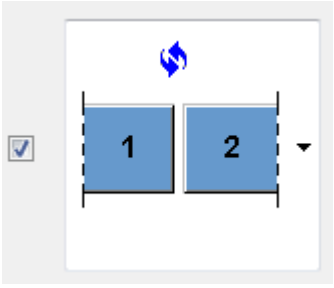
Situation	Description
	Wall to wall connection with groove, shear teeth, U bars and a loose bar.

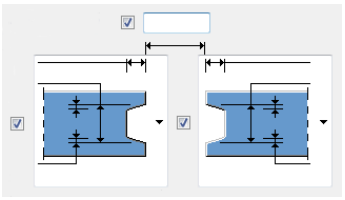
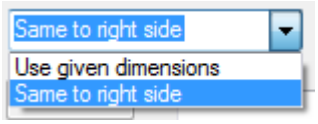
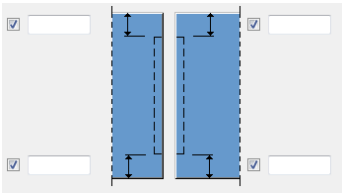
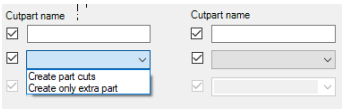
Selection order

1. Select the first wall.
2. Select the second wall.

   The connection is created automatically when the second wall is selected.

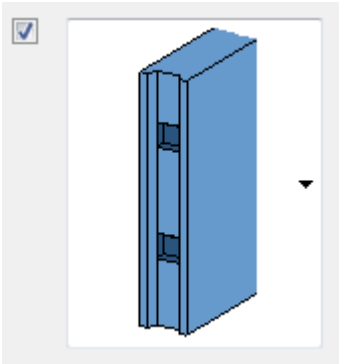
Edge shape tab
Use the **Edge shape** tab to select the wall framing type

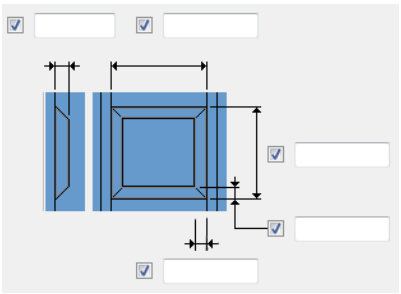
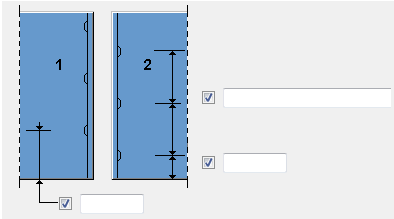
Option	Description
	Wall framing type selection. Wall framing type selection affects the wall edge shape options. You can use the connection in three different modeling situations: • end-to-end • corner With this option the first selected wall is fitted to align with the exterior face of the second selected wall. • tee

Option	Description
	Wall edge shape. Select the edge shape of each wall. Adjust the free space between two walls.
	Wall edge dimensions. To set the edge shape and dimensions independently for both walls, use the **Use given dimensions** option. To make the edge shape similar in both walls, use the **Same to right side** option.
	Groove start and end point. You can define the start point and the end point of the groove. If you do not enter a value, or enter `0`, the groove is applied at full height of the wall.
	Select to create part cuts or only extra part cuts, and define the name of the cut. You can set this separately for both walls. For part properties of extra part cuts, select a saved settings file.

Extra teeth tab

Use the **Extra teeth** tab to define the properties for additional shear teeth.

Option	Description
	Select whether additional shear teeth or shear cups are created. Teeth or cups can also be created when there is no groove.

Option	Description
	Dimensions of a single tooth or cup.
	Location of the first tooth or cup in the first wall and in the second wall, and the spacing between the teeth or cups. You can enter a list of spaces, for example `10*400`, or just a single value. When you enter a single value, the number of teeth or cups is calculated automatically based on the height of the connection.
	Select to create part cuts or only extra part cuts, and define the name of the cut.

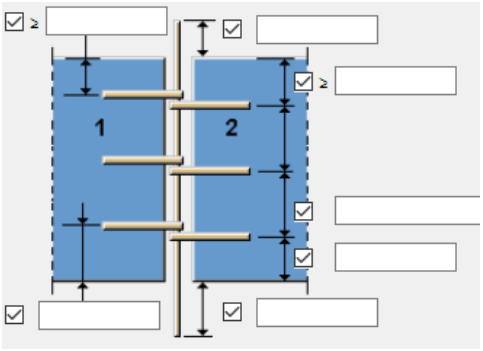
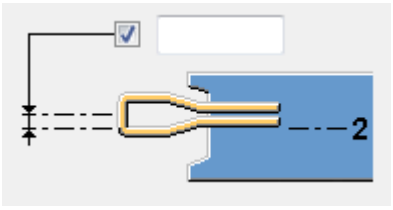
Connectors tab
Use the **Connectors** tab to select the type of connectors created.

Connector type

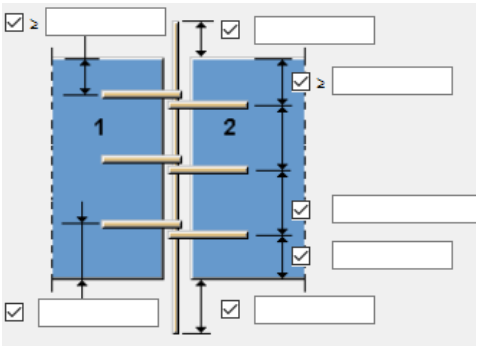
The content of the **Connectors** tab varies depending on the type of the selected connectors.

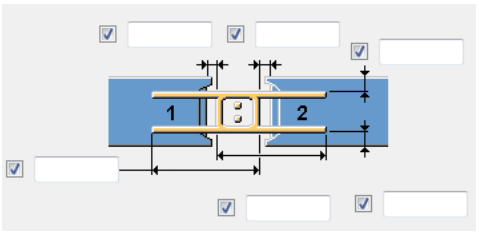
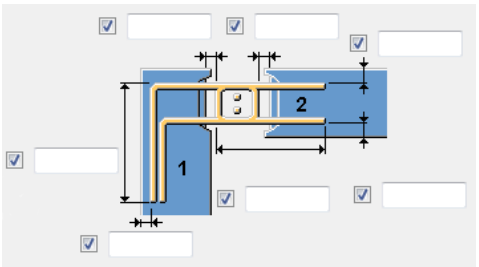
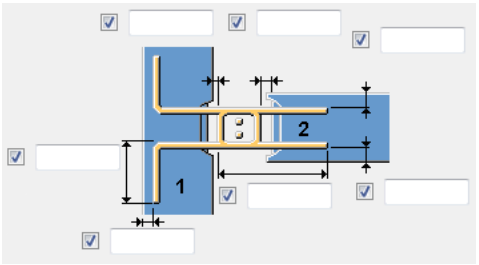
Option	Description
	Connector type. • No connectors. Only groove with or without shear teeth is created. • Connectors are created by using the selected embed (custom part). • Connectors are created by adding reinforcing bar loops. Reinforcing bar loop is formed depending on the wall framing type. In addition, the connector can optionally create one or more long reinforcing bars along the connection.

Embeds and loose rebar

Option	Description
	Location of the first connector (embed) in the first wall and in the second wall, location of the last rebar/embed in the wall top edge, and the spacing between the connectors. You can enter a list of spaces, for example `10*400`, or just a single value, for example `300`. When you enter a single value, the number of connectors is calculated automatically based on the height of the connection. You can also define the extended length of the loose long bar.
	Select which embeds (custom parts) are created in each wall. If the embed has any saved attributes, select which attributes are used.
	Offset of the embed. By default, the embed input points are in the center of the wall.

U bars and loose rebar

Option	Description
	Location of the first connector (reinforcing bar loop) in the first wall and in the second wall, and the spacing between the connectors. You can enter a list of spaces, for example `10*400`, or just a single value, for example `300`. When you enter a single value, the number of connectors is calculated automatically based on the height of the connection. You can also define the extended length of the loose long bar.

Option	Description
  	Reinforcing bar loop dimensions. The available dimension options depend on the wall framing type selected on the **Edge shape** tab. Many of the dimensions have counter parts in the first wall and in the second wall. It is enough to enter one dimension and by default the counter part dimension gets the same value.
Number of bars	Number of long loose bars. If you do not want to create a loose long bar, enter `0` as the **Number of bars**.

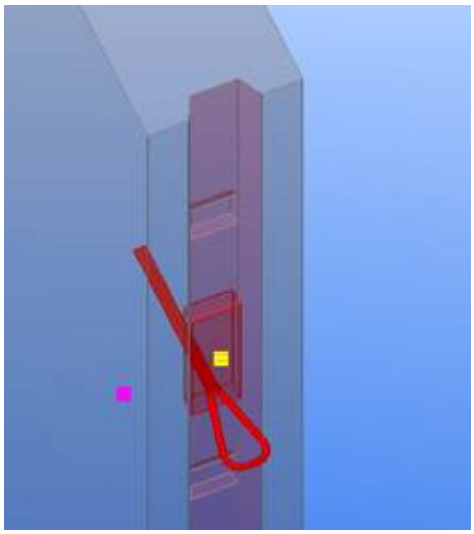
Reinforcing bar properties

Enter the properties for reinforcing bar **Grade**, **Size**, **Name**, **Class**, **Prefix** and **Start Number**.

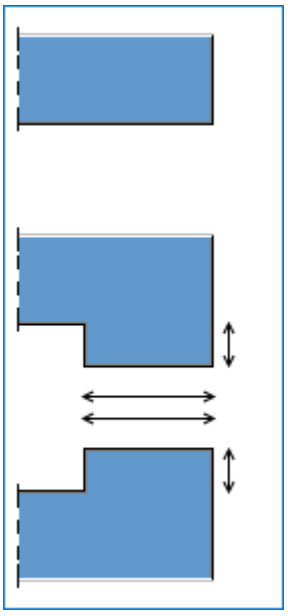
Customization

If needed, you can create your own embed (custom part) and use it as a connector.

When creating the embed, ensure that the input points are defined so that they are compatible with **Wall to wall connection**. The image below shows examples of the input points.



Thickening tab
Use the **Thickening** tab to create an extra thicknening to one or both walls.

Option	Description
	Select to which side of the wall the thickening will be created. The final location of the extra thickening depends on the actual framing case (end to end, L corner, T corner). Enter the name and class for the extra thickening. Leave the input box empty to use the wall name and/or class.

Wall groove seam detail

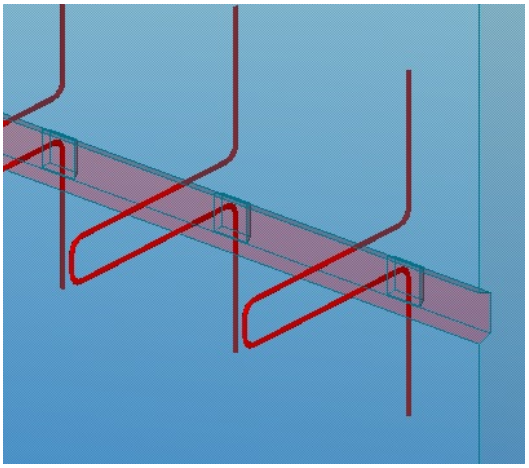
Wall groove seam detail creates a groove with optional shear teeth to a precast wall, a slab or a similar part. Additionally the connection can create reinforcing bar loops or embeds that work as connectors in the walls.

Objects created

- Groove

- Shear teeth or shear cups
- U bars
- Embeds
- Long bar

Use for

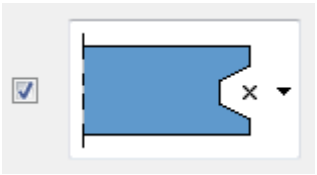
Situation	Description
	Groove with shear teeth and U bars.

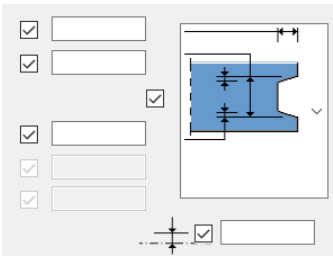
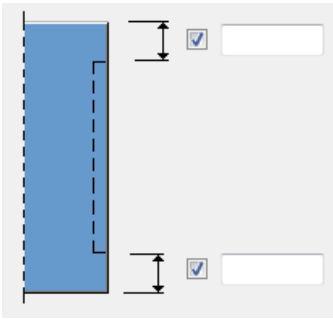
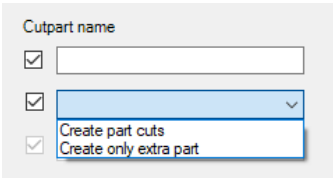
Selection order

1. Select the wall or slab.
2. Pick the start point of the groove.
3. Pick the end point of the groove.

Edge shape tab

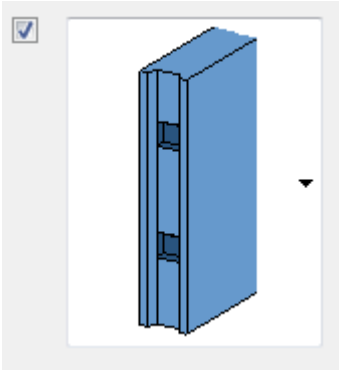
Use the **Edge shape** tab to select the orientation of the groove.

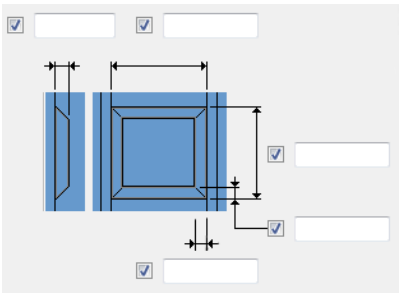
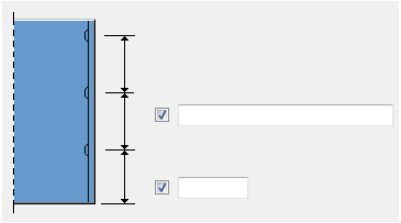
Option	Description
	Groove orientation selection. Define the orientation of the groove in relation to the input points.
Create fitting	Select whether the part end fitting is created.

Option	Description
	Groove edge shape.
	Groove start and end point in relation to input points. You can define the start point and the end point of the groove. If you do not enter a value, or enter `0`, the groove is applied at full height of the wall.
	Select to create part cuts or only extra part cuts, and define the name of the cut. For part properties of extra part cuts, select a saved settings file.

Extra teeth tab

Use the **Extra teeth** tab to define the properties for additional shear teeth.

Option	Description
	Select whether additional shear teeth or shear cups are created. Teeth or cups can also be created when there is no groove.

Option	Description
	Dimensions of a single tooth or cup.
	Location of the first tooth or cup in the groove, and the spacing between the teeth or cups. You can enter a list of spaces, for example `10*400`, or just a single value. When you enter a single value, the number of teeth or cups is calculated automatically based on the height of the detail.
	Select to create part cuts or only extra part cuts, and define the name of the cut.

Connectors tab

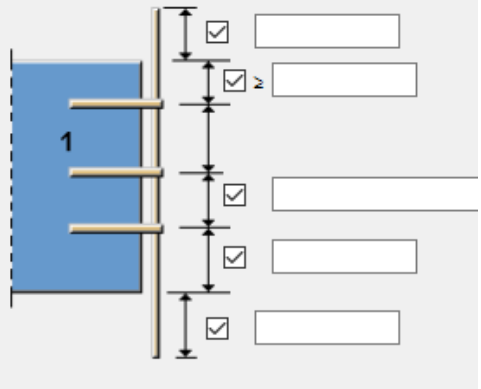
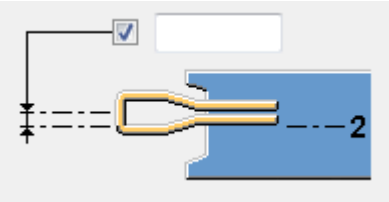
Use the **Connectors** tab to select the type of connectors.

Connector type

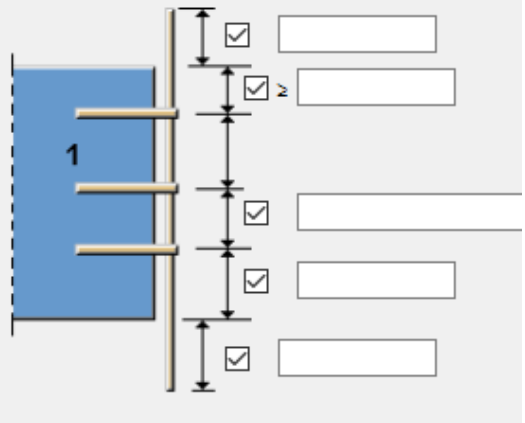
The content of the **Connectors** tab varies depending on the type of the selected connectors.

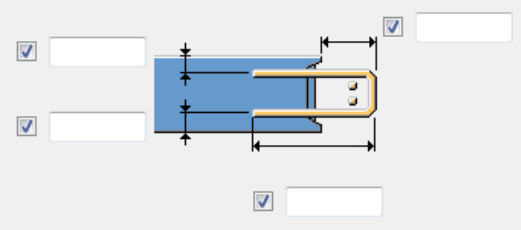
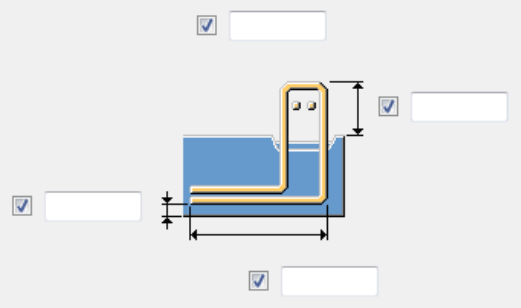
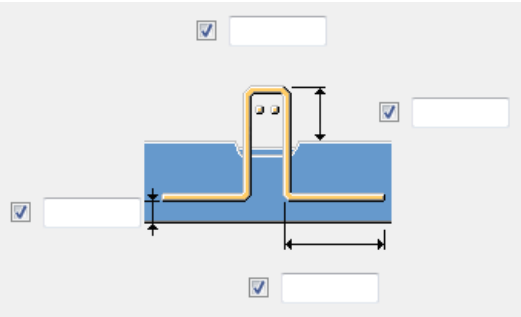
Option	Description
	Connector type. • No connectors. Only groove with or without shear teeth is created. • Connectors are created by using the selected embed (custom part). • Connectors are created by adding reinforcing bar loops. Reinforcing bar loop is formed depending on the framing type. In addition, the connector can optionally create one or more long reinforcing bars along the connection.

Embeds and loose rebar

Option	Description
	Location of the first connector (embed) in the groove, location of the last rebar/ embed in the wall top edge, and the spacing between the connectors. You can enter a list of spaces, for example `10*400`, or just a single value, for example `300`. When you enter a single value, the number of connectors is calculated automatically based on the height of the detail. You can also define the extended length of the loose long bar.
	Select which embeds (custom parts) are created in each groove. If the embed has any saved attributes, select which attributes are used.
	Offset of the embed. By default, the embed input points are in the center of the groove.

U bars and loose rebar

Option	Description
	Location of the first connector (reinforcing bar loop) in the groove, and the spacing between the connectors. You can enter a list of spaces, for example `10*400`, or just a single value, for example `300`. When you enter a single value, the number of connectors is calculated automatically based on the height of the detail. You can also define the extended length of the loose long bar.

Option	Description
  	Reinforcing bar loop dimensions. The available dimension options depend on the orientation of the groove selected on the **Edge shape** tab.
Number of bars	Number of long loose bars If you do not want to create a loose long bar, enter `0` as the **Number of bars**.

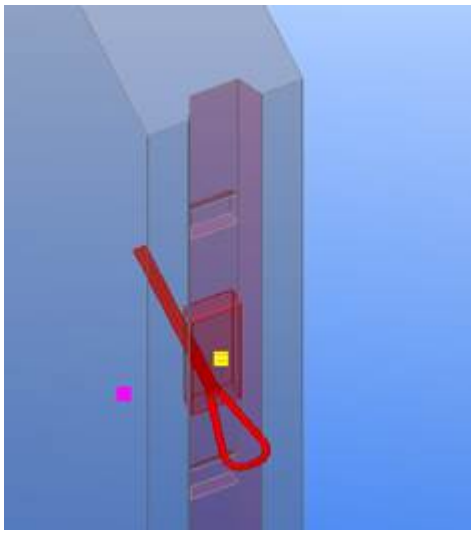
Reinforcing bar properties

Enter the properties for reinforcing bar **Grade**, **Size**, **Name**, **Class**, **Prefix** and **Start Number**.

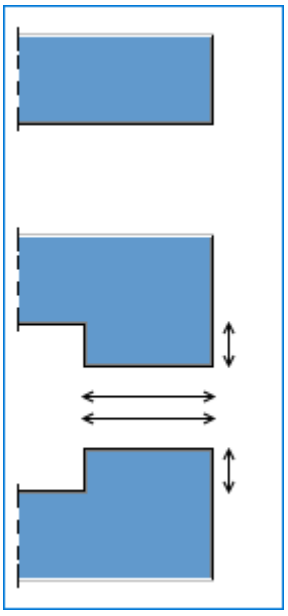
Customization

If needed, you can create your own custom part (embed) and use it as a connector.

When creating the embed, ensure that the input points are defined so that they are compatible with **Wall groove seam detail**. The image below shows examples of the input points.



Thickening tab
Use the **Thickening** tab to create an extra thicknening to the wall.

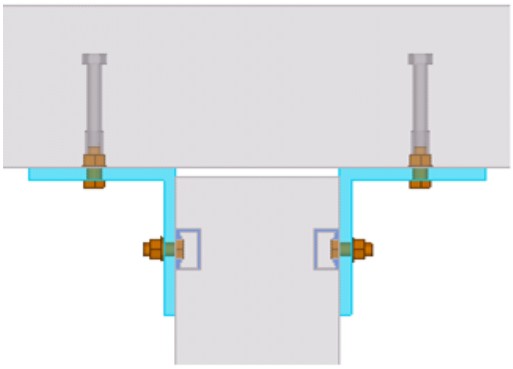
Option	Description
	Select to which side of the wall the thickening will be created. The final location of the extra thickening depends on the actual framing case (end to end, L corner, T corner). Enter the name and class for the extra thickening. Leave the input box empty to use the wall name and/or class.

Anchor (10)
Anchor (10) connects two precast concrete parts, for example, two panels or a column to a panel. The precast concrete parts are connected with L profiles and with embedded connection profiles that act as anchors, for example, custom component bolt anchors or cast-in channels. The bolt anchors and the cast-in channels can be single-sided or double-sided. Additionally, seams can be defined between the main part and the secondary parts.

Objects created

- Anchors (bolt anchors or cast-in channels)
- L profiles
- Bolts

Use for

Situation	Description
	Connection with L profiles and bolt anchors.
	Connection with L profile and cast-in channel.

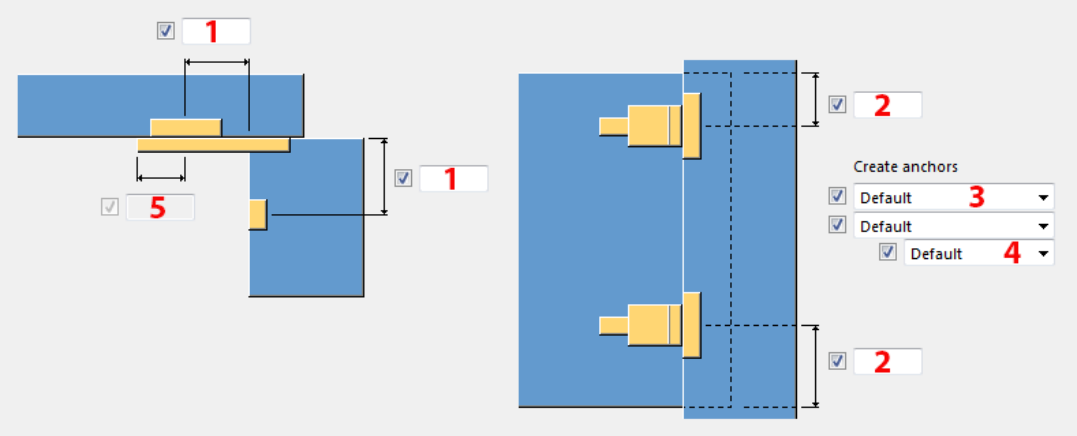
Selection order

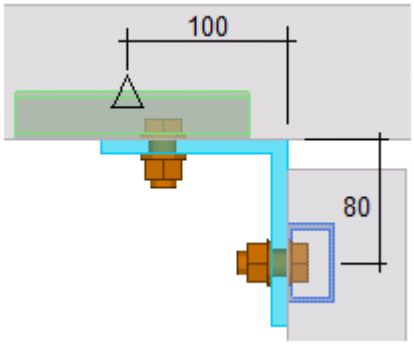
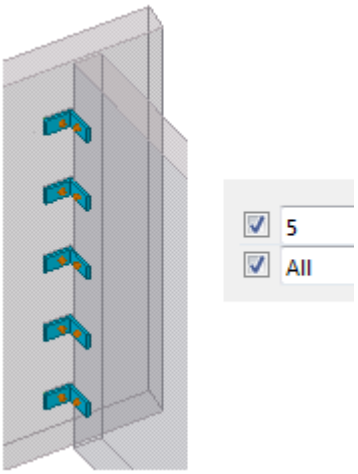
1. Select the main part (precast concrete panel or a column).
2. Select the secondary part (precast concrete panel).

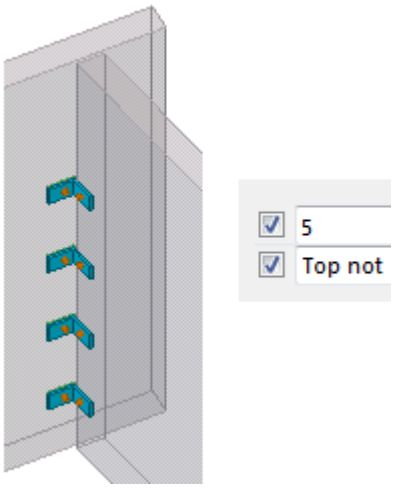
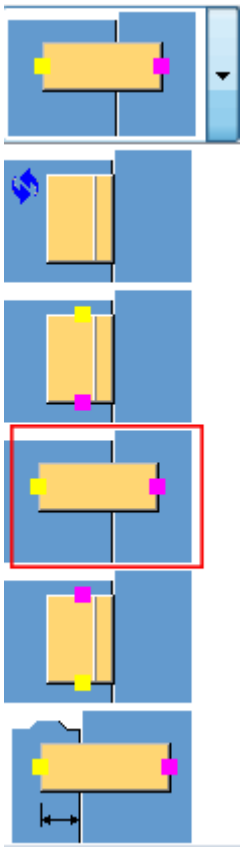
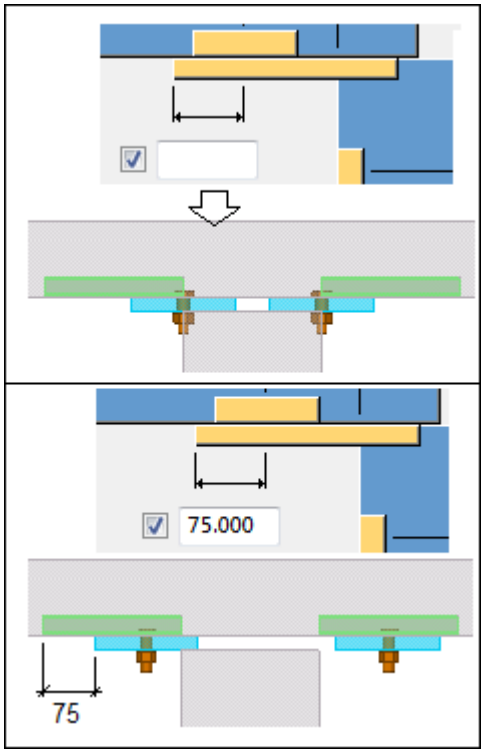
   The connection is created automatically when the secondary part is selected.

Picture tab

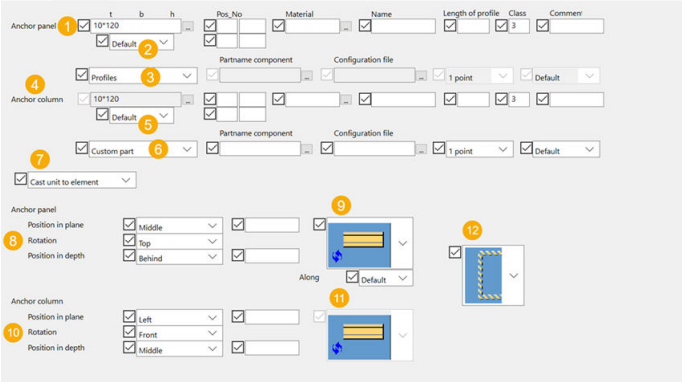
Use the **Picture** tab to define the number of anchors (bolt anchors or cast-in channel profiles), edge distances and plate embed offset.

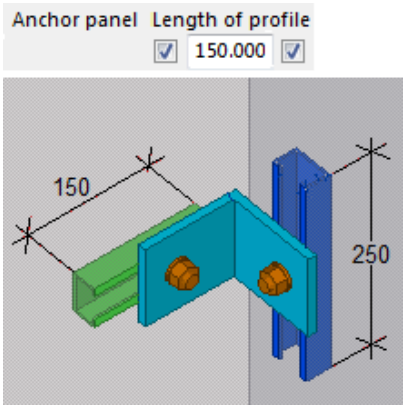
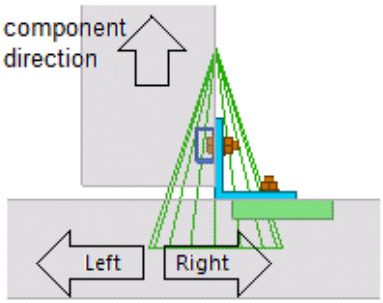


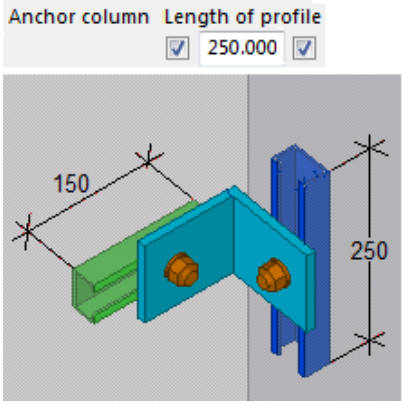
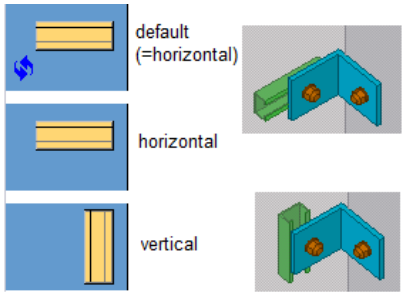
	Description	Example
1	Horizontal offset of the anchors. The offset can be defined for both concrete parts separately.	
2	Vertical offset of the anchors. The offset can be defined for the top and bottom side.	
3	Select how the anchors are distributed.	

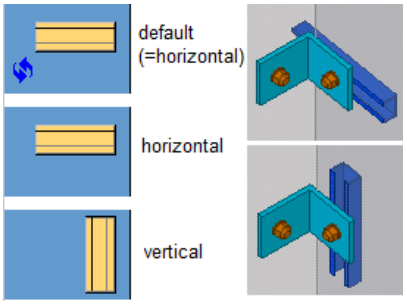
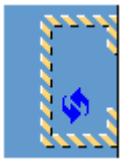
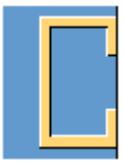
	Description	Example
4	Select which of the anchors are omitted.	
5	Connection plate horizontal offset from the centerline of an anchor. This option is available only if you have set the orientation of the connection plate as follows on the **Conn profile** tab: 	

Anchors tab
Use the **Anchors** tab to define the properties, position and orientation of the anchors (bolt anchors or cast-in channel profiles).



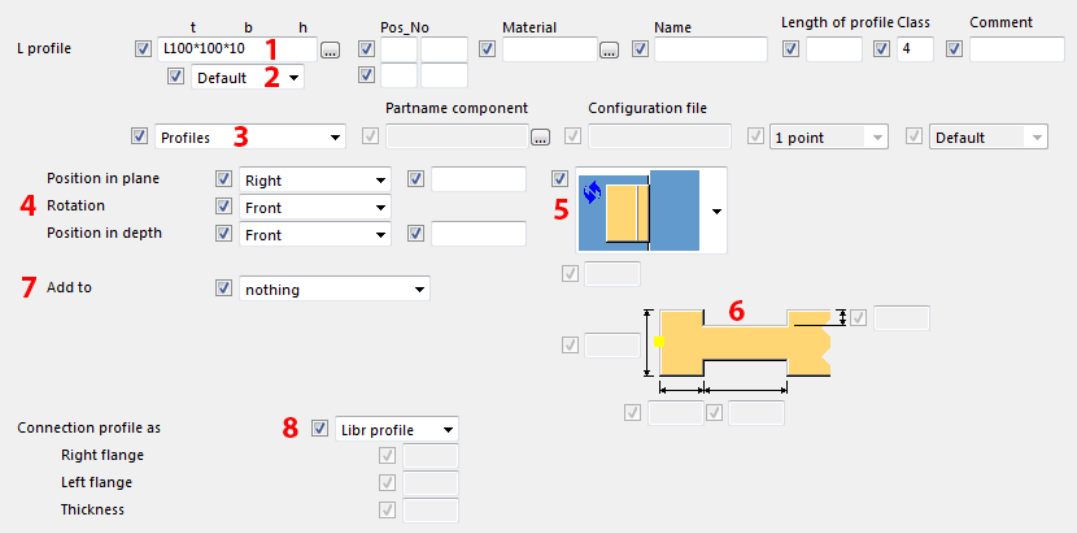
	Description	Example
1	Anchor properties in the main part.	
2	Select on which side the anchors are created in the main part.	
3	Select whether a profile from the profile catalog is used as an anchor, or a custom part from the **Applications & components** catalog.	

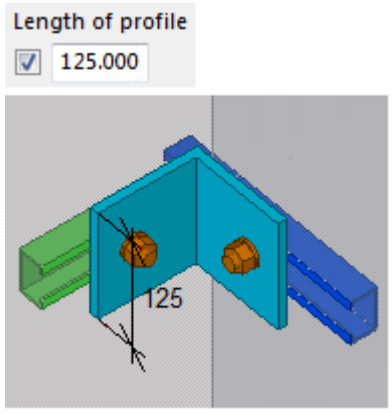
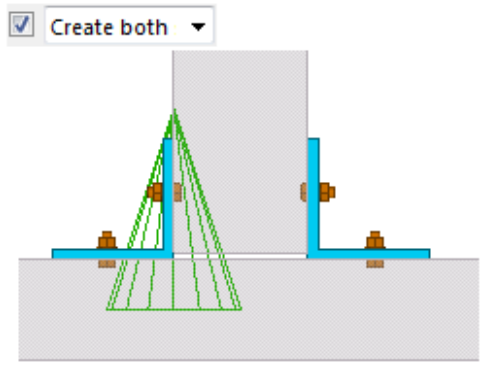
	Description	Example
4	Anchor properties in the secondary part.	
5	Select on which side the anchors are created in the secondary part.	
6	Select whether a profile from the profile catalog is used as an anchor, or a custom part from the **Applications & components** catalog. To use saved custom component properties, select the saved properties file.	
7	Select the connection method of the anchor. • **Default** (same as **Weld to element**) • **Weld to element** • **Cast unit to element** • **Not add to element** (bolt anchors or cast-in channels are loose parts) • **Sub-assembly**	
8	Position and rotation of the anchors in the main concrete part. For the **Position in plane** and **Position in depth** options, an offset can be defined.	
9	Orientation of the anchors in the main part.	

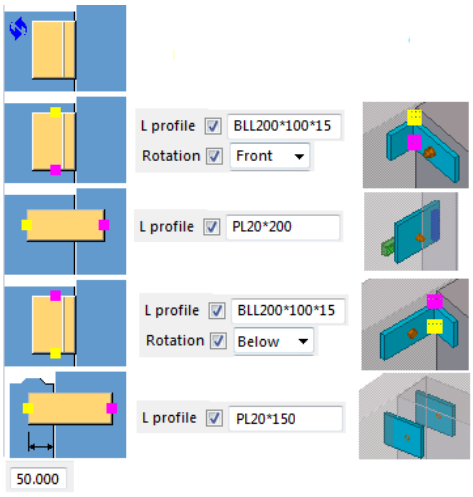
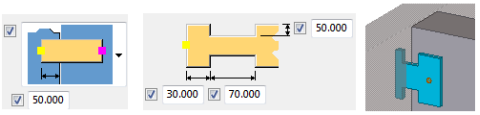
	Description	Example
10	Position and rotation of the anchors in the secondary concrete part. For **Position in plane** and **Position in depth** options, an offset can be defined.	
11	Orientation of the anchors in the secondary part.	
12	Select the cut-out method of the anchors.	 Default, no cut
		 No cut
		 Cut-out shape is the exact profile contour of the bolt anchor or the cast-in channel
		 Box shape cut-out

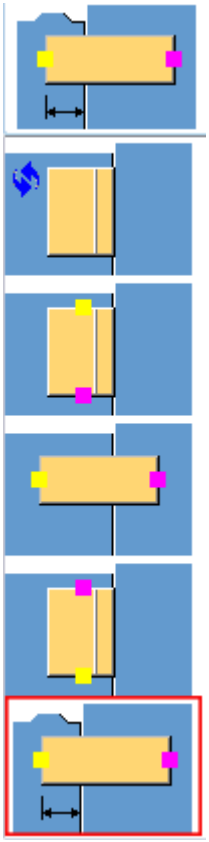
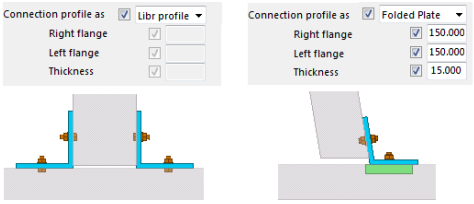
Conn profile tab

Use the **Conn profile** tab to define the properties and position of the connection L profile.



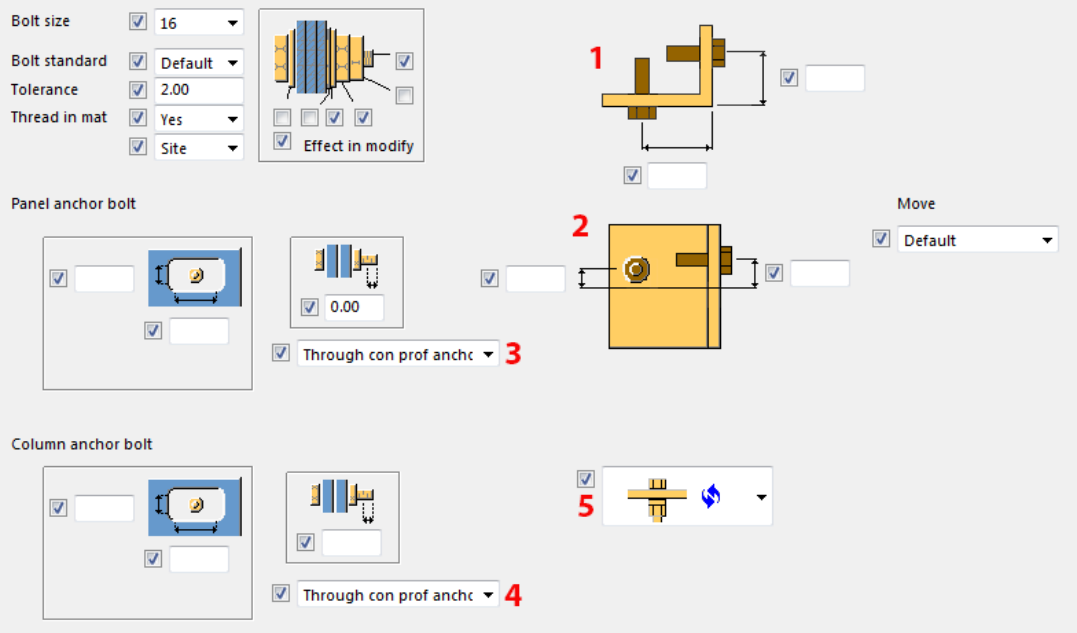
	Description	Example
1	Connection profile properties.	
2	Select on which side the connection profiles are created.	
3	Select whether a profile from the profile catalog is used as a connection profile, or a custom part	

	Description	Example
	from the **Applications & components** catalog.	
4	Position and rotation of the connection profiles. For the **Position in plane** and **Position in depth** options, an offset can be defined.	
5	Orientation of the connection profile.	
6	Connection profile cuts. These options are available only if you have set the orientation of the	

	Description	Example
	connection profile as follows on the **Conn profile** tab: 	
7	Select the connection method of the connection profile.	
8	Select the connection profile type. • **Library profile** (define the profile using the L profile options). • **Folded plate** (define the dimensions using the options below).	

Bolts tab

Use the **Bolts** tab to define the bolt properties and offsets.



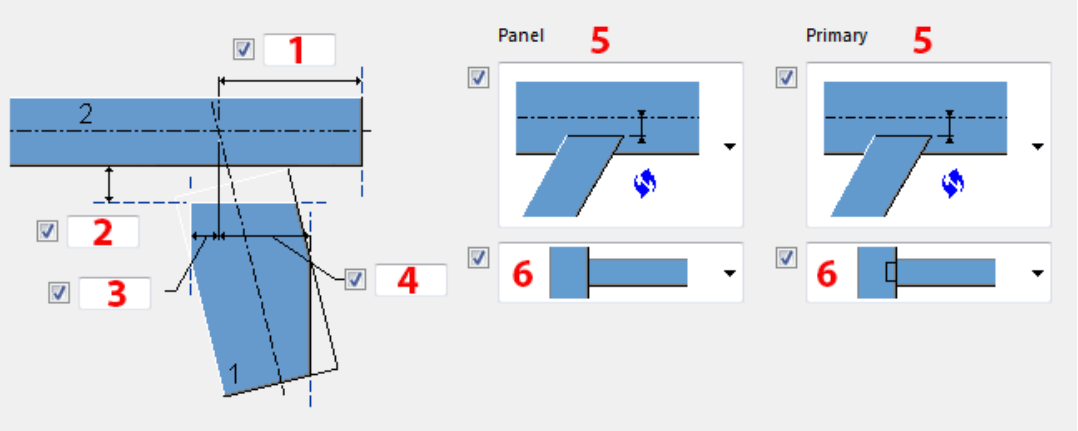
	Description
1	Horizontal bolt distances from the L profile edges. Default value is half of L profile flange width.
2	Vertical bolt offsets. Reference is the L profile centerline. Default value is 0 mm.
3	Select how the bolts are added in the main part.
4	Select how the bolts are added in the secondary part.
5	Bolting direction.

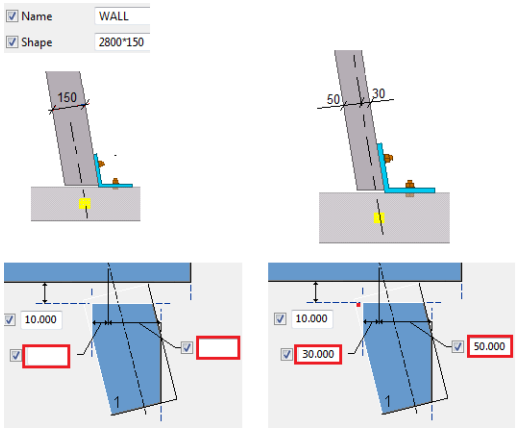
Cuts tab

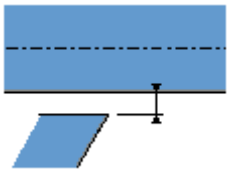
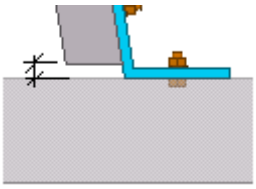
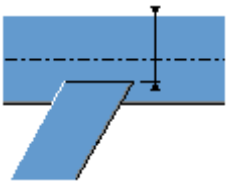
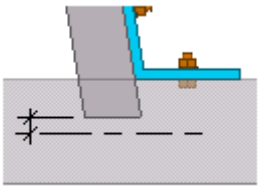
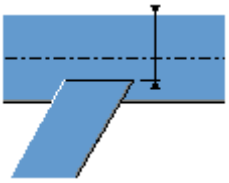
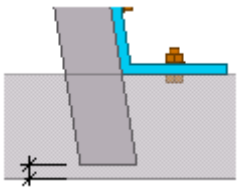
Use the **Cuts** tab to define the cuts and the seams in the main and secondary parts.

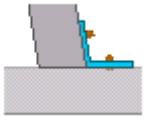
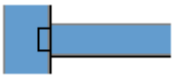
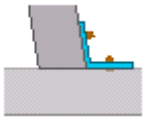
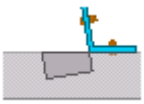
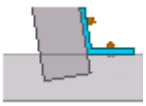
Additionally, the main part can be extended, or the secondary part thickness can be increased or decreased.

You can create a seam by defining a cut between the main part and the secondary part.



	Description	Example
1	Extension of the main part. If you need to extend the part, enter a value. Ensure that you have also selected a cut method from the cut type list. If you select the **No cut** option, the part is not extended.	
2	Seam width. Reference is the L profile centerline. Default value is 0 mm.	
3	Left side cut along the secondary part. Reference is the part centerline. Part thickness is decreased.	
4	Right side cut along the secondary part. Reference is the part centerline. Part thickness is decreased.	

	Description	Example
5	Cut reference for the main and secondary parts.	 Main part near side   Main part centerline   Main part far side 
6	Select the cut type.	 Default, no cut

	Description	Example
		 Reference is the main part. Secondary part will be adjusted to the main part using a fitting. 
		 Reference is the main part. Secondary part will be adjusted to the main part using a line cut. 
		 Reference is main part. Just like previous options, but the longest part of the panel will be removed using a line cut. 
		NO CUT No cut 

Default tab
Use **Default** tab to define part distances.

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

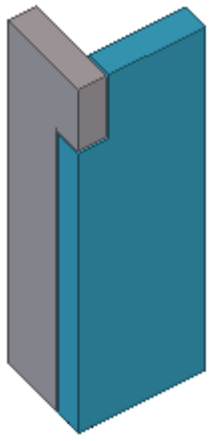
Wall wall teeth (12)

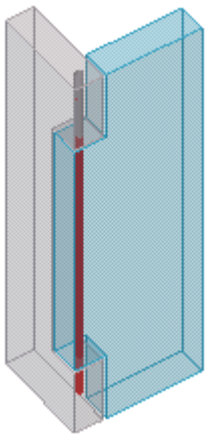
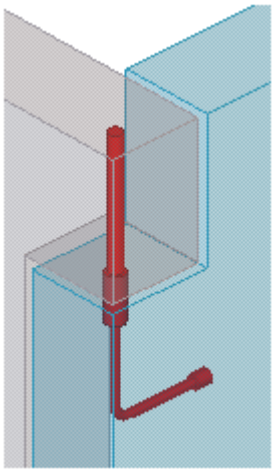
Wall wall teeth (12) connects two perpendicular concrete panels with a tooth-shaped connection. The number of teeth can be set. Injection tubes and connection bars can be added to the connection. The connection is created with or without seams.

Objects created

- Tooth connection (2 or 3 teeth) between two panels
- Injection tubes
- Connection bars

Use for

Situation	Description
	2-teeth connection between two panels. The height of the teeth can be modified and the clearance between the teeth can be set.

Situation	Description
	3-teeth connection between two panels with connection bar.
	Tooth connection between two panels with connection bar and socket.

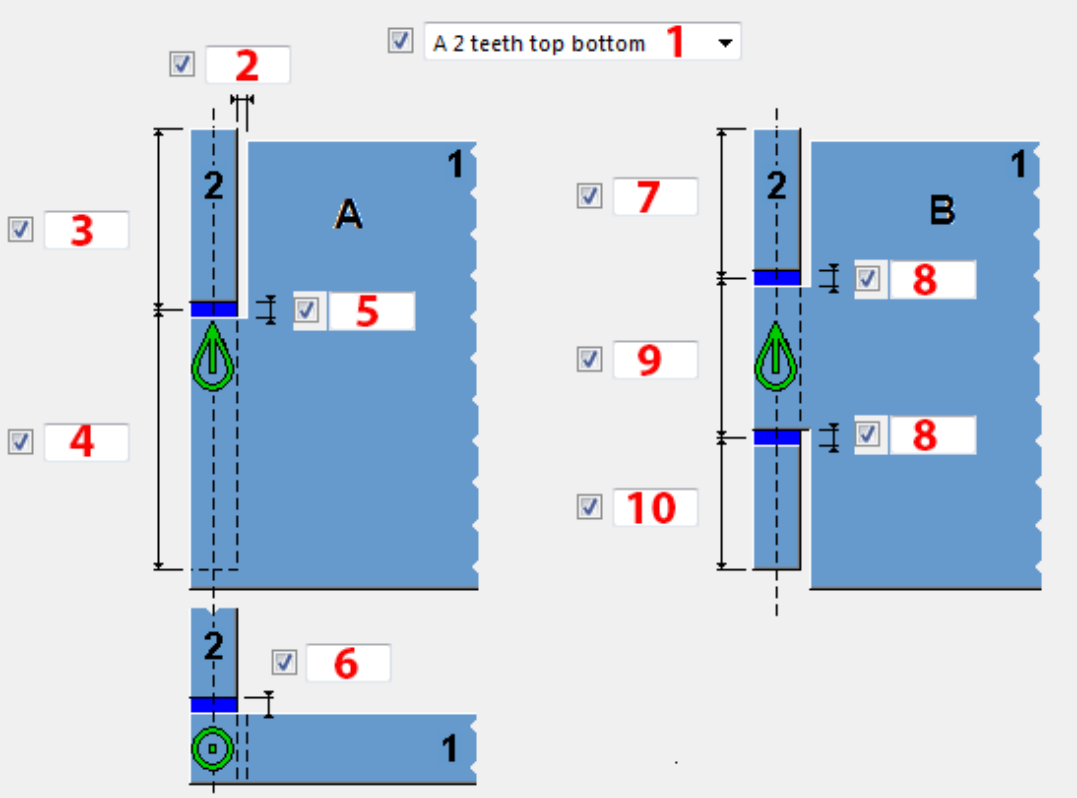
Selection order

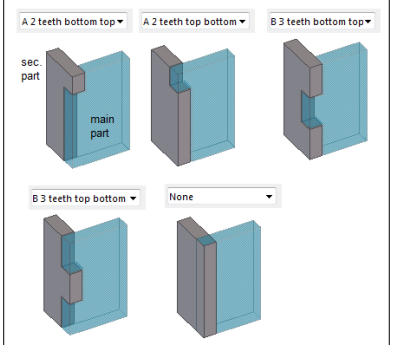
1. Select the main part (concrete panel).
2. Select the secondary part (concrete panel).

   The connection is created automatically when the secondary part is selected.

Picture tab

Use the **Picture** tab to define the number of teeth, the height of teeth, and seam widths. **Wall wall teeth (12)** has two main connection types: **A** (two teeth) and **B** (three teeth).

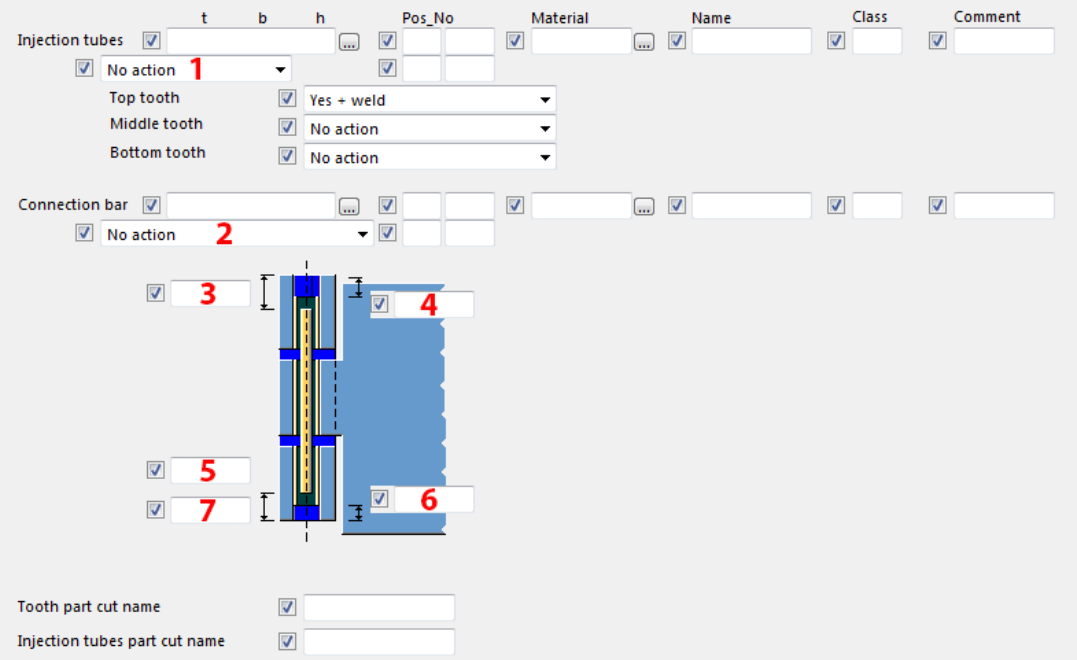


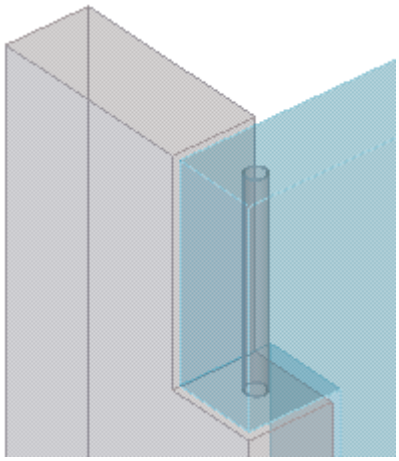
	Description
1	Select the number and position of teeth. **Standard** is **A 2 teeth bottom up**. 
2	Horizontal seam width in the main part. For type **A** (2-teeth connection).
3	Secondary part tooth height. Reference is center of the seam. For type **A** (2-teeth connection).
4	Main part tooth height. Reference is center of the seam. For type **A** (2-teeth connection). This value has an effect only if the secondary part tooth height (**3**) has no value.
5	Vertical seam width between the teeth. For type **A** (2-teeth connection).
6	Horizontal seam width in the main part. For type **A** (2-teeth connection).

	Description
7	Top tooth height. For type **B** (3 teeth connection). Reference is center of the seam. If there is no value, and the middle tooth height and the bottom tooth height are defined, then the top tooth gets the residual height distance. If none of the tooth heights are defined, then all three teeth get an equal height: `(wall height - seam width) / 3` This value has lowest priority from the three tooth height-defining values. Example: Panel height is 1500mm. All three tooth heights are defined: • Top: 400mm • Middle: 700mm • Bottom: 600mm Total defined height of teeth = 1800mm. Result: top tooth gets height 1500-700-600=200mm
8	Vertical seam width between the teeth. For type **B** (3-teeth connection).
9	Bottom tooth height. Reference is center of the seam. For type **B** (3-teeth connection). If there is no value, and the top tooth and the middle tooth height are defined, then the bottom tooth gets the residual height distance.

Parts tab

Use the **Parts** tab to define the properties and dimensions of injection tubes and connection bars.



Option	Description
Injection tubes	Injection tube properties. You can define the placing of the injection tubes separately for the top tooth, middle tooth and bottom tooth.
1	Select whether an injection tube is created. Example:  Default is **No action**.
Top tooth **Middle tooth** **Bottom tooth**	Select whether an injection tube is created in the teeth and select the connection method. Default is **Part cut**.

Option	Description
Connection bar	Connection bar properties. One connection bar can be created through the teeth.
2	Select whether a connection bar is created and the connection method. Default is **Yes + weld 1**.
3	Depth of a connection bar from the top of the top tooth. Default is **0**.
4	Depth of an injection tube from the top of the top tooth. Default is **0**.
5	Depth of a connection bar from the top of the bottom tooth. Default is **top tooth height + seam width**. If both values **5** and **7** are defined, then the value in the box **7** overwrites the value in the box **5**.
6	Depth of an injection tube from the bottom of the bottom tooth. Default is **0**.
7	Depth of a connection bar from the bottom of the bottom tooth. If both values **5** and **7** are defined, then the value in the box **7** overwrites the value in the box **5**.

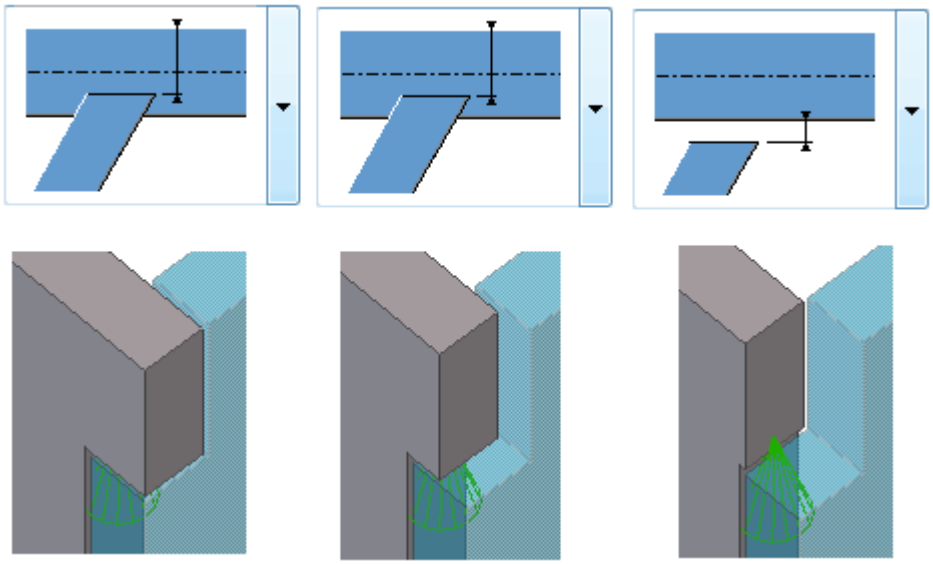
Cut sec tab

Use the **Cut sec** tab to define the way the secondary concrete panel is cut.

By default the component creates a 2-teeth connection.

You can also define a fixed offset value for the cut.

If the concrete panels are not perpendicularly orientated, you can define how the secondary part is fitted to the main part.

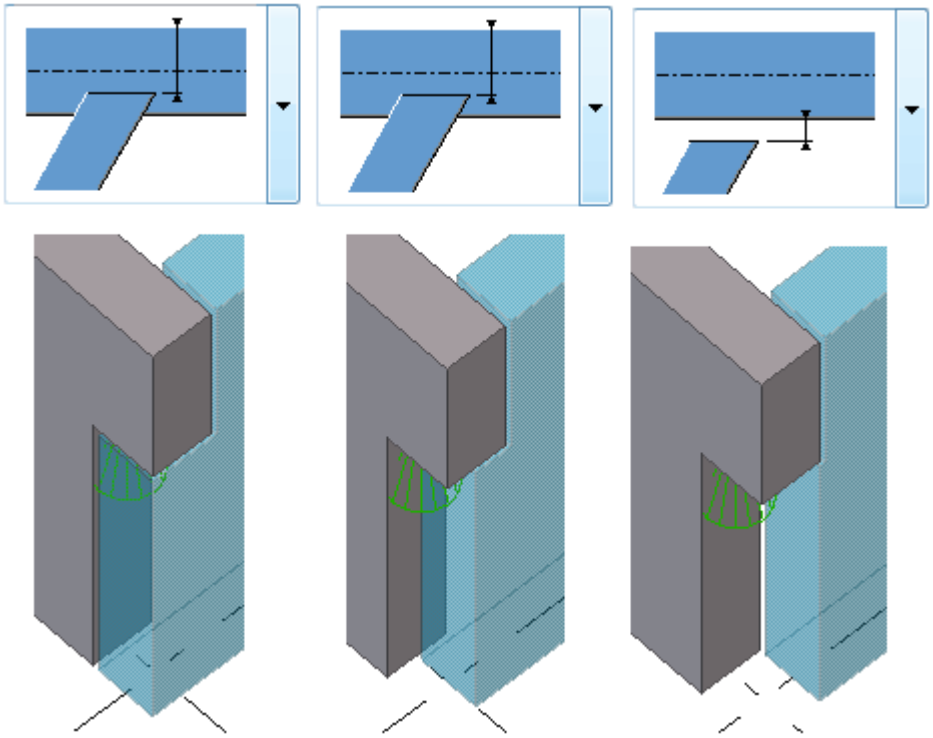


Cut prim tab

Use the **Cut prim** tab to define the way the main concrete panel is cut.

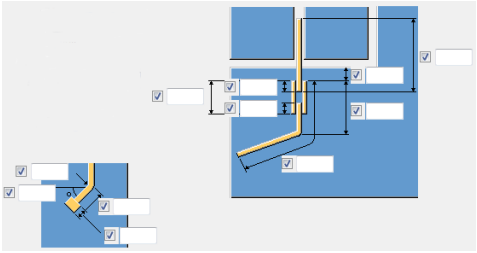
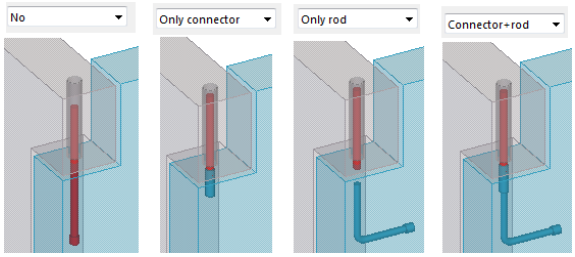
You can also define a fixed offset value for the cut.

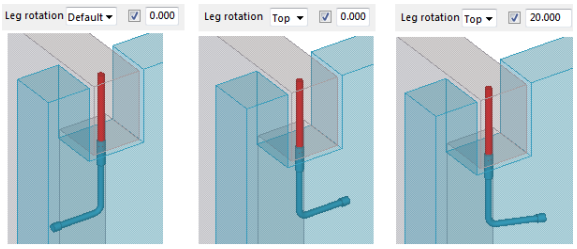
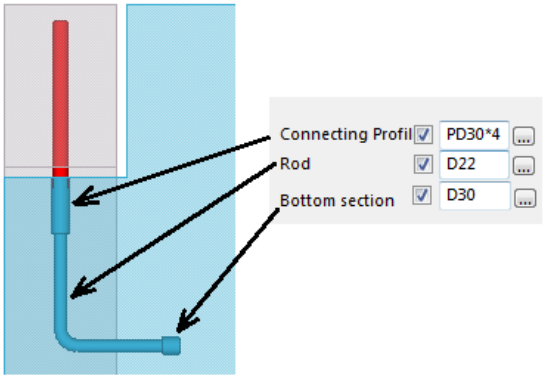
If the concrete panels are not perpendicularly orientated, you can define how the main part is fitted to the secondary part.



Socket tab

Use the **Socket** tab to define whether socket anchors are created, how they are connected and the dimensions and position of the socket anchors. Sockets are created only if you have created connection bars.

Option	Description
	Dimensions for socket anchor, rod profile and connecting profile.
Create socket	Select whether sockets are created and which parts are included. Note that a socket is created only if a connection bar has been created on the **Parts** tab. In the examples the red profile is the connection bar: 
Connect socket	Select how the sockets are connected to the main part. Default is **By cast unit**.
Connection rod-connector	Select the connection method between the rod and the connecting profile. Default is **Part add conn to rod**.
Rod type	Select the rod type. Default is **Poly-profile**.
Leg rotation	Select the direction of the rod. You can enter an angle in the box on the right.

Option	Description
	Default is **Front**. 
Connecting profile **Rod** **Bottom section**	Properties for the connecting profile, rod and bottom section. Note that the bottom section is part-added to the rod automatically. Example: 
Partname component	If you want to use a custom part to create the socket, select the **Custom part** option in the **Create socket** list. Then browse for the component, and use the list of options to position the custom part. To use saved custom component properties, select the saved properties file.

General tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

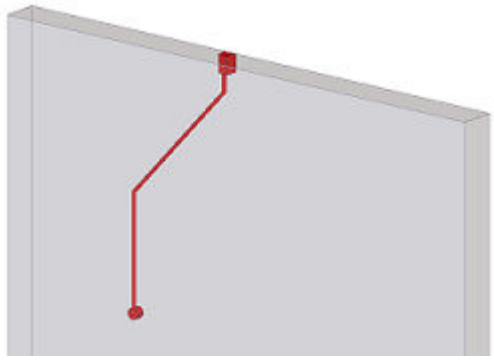
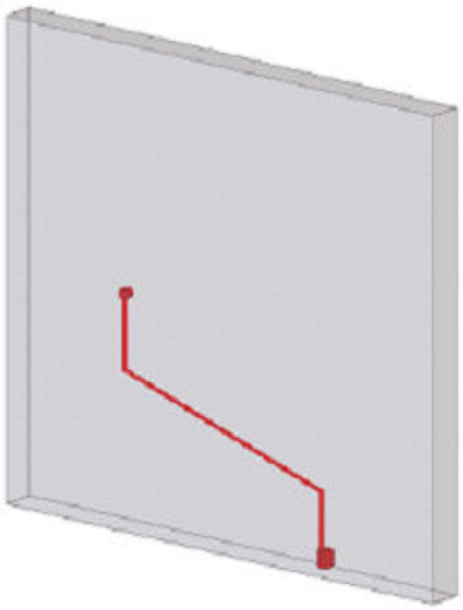
Electric box in wall (84)

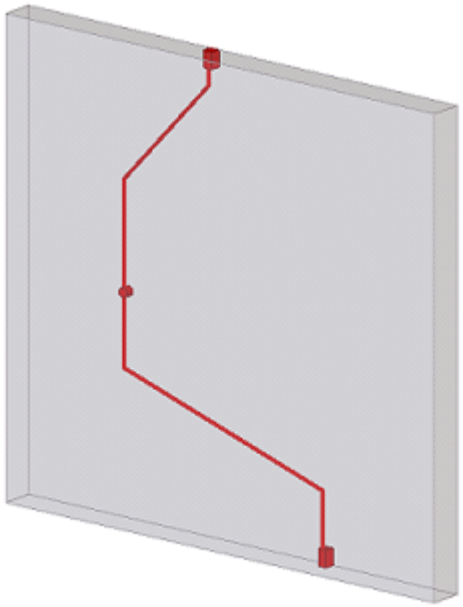
Electric box in wall (84) creates electric boxes in walls.

Objects created

- Electric boxes
- Tubes

Use for

Situation	Description
	Electric box in a wall with top connection.
	Electric box in a wall with bottom connection.

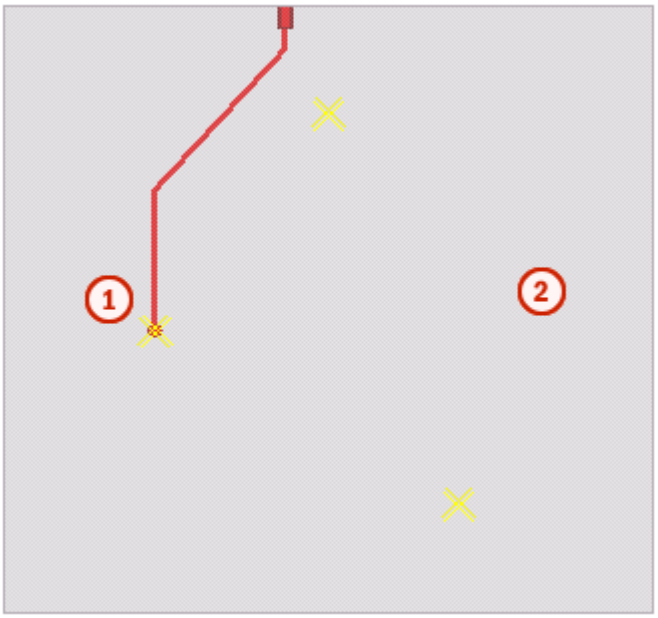
Situation	Description
	Electric box in a wall with top and bottom connections.

Selection order

1. Select a concrete panel.
2. Pick a position for the electric box.
3. Pick a position for the top connection.
4. Pick a position for the bottom connection.

   The electric box and connections are created automatically.

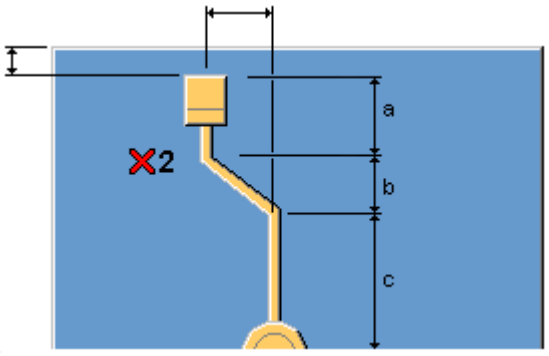
Part identification key

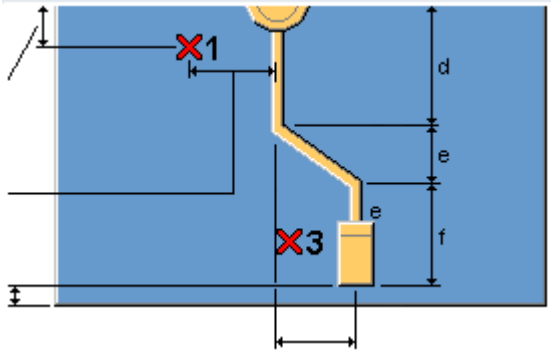
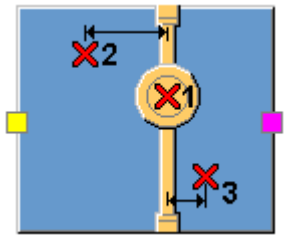
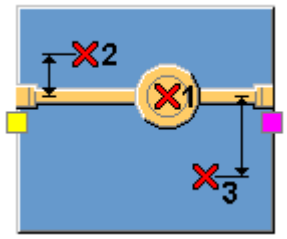


	Part
1	Electric box
2	Panel

Picture tab
Use the **Picture** tab to control the shape, number and position of the tubes and the position of the electric boxes.

Tube direction

Option	Description
	Select the tube shape in the up direction.

Option	Description
	Select the tube shape in the down direction.
 	Select the tube direction in relation to the start point (yellow) and end point (magenta) of the wall.
	Select the number of tubes in the up direction.
	Select the number of tubes in the down direction.

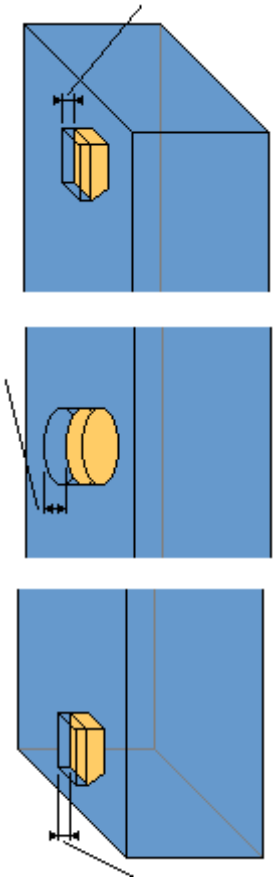
Tubes to concrete / Electric parts to concrete

Select how the tubes and the electric parts are connected to the concrete part.

Option	Description
As subassembly	Add as a subassembly to the concrete part.
By cast unit	Add to the concrete part.

Option	Description
Welds	Weld to the concrete part.
Not	Do not connect to the concrete part.

Tube and electric box positions

Option	Description
	Select the position of the up connection in the concrete part. Select the position of the electric box in the concrete part. Select the position of the down connection in the concrete part.

Parts tab

Use the **Parts** tab to control the tube properties at the top and bottom of the concrete part and the position of the tube reference points.

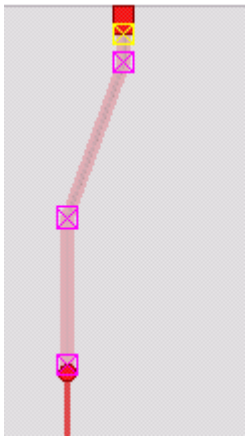
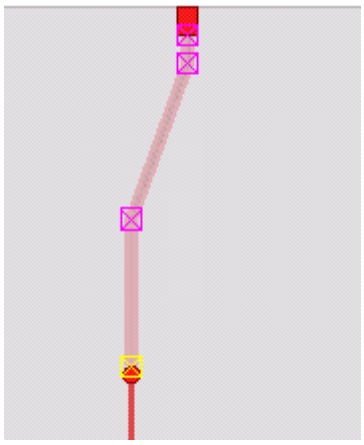
Tube top / Tube bottom

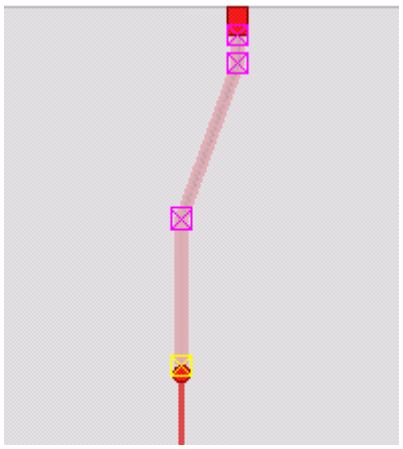
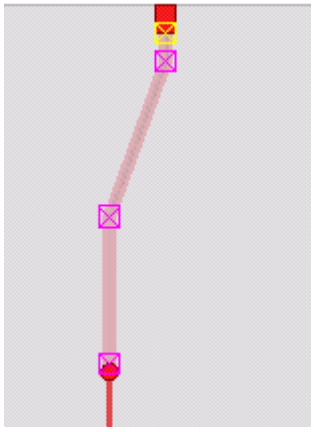
Option	Description	Default
t, **b**, **h**	Define the tube thickness, width and height.	`D19`
Pos_No	Define a prefix and a start number for the part position number.	`ET 1`

Option	Description	Default
Material	Define the material grade.	PVC
Name	Define a name for the tube.	E-tube
Class	Define the part class number for the tube.	2
Comment	Add a comment for the tube.	

1st reference point

Define a reference point for the tube. The reference point determines the position of the part mark in a drawing.

Option	Description	Example
At top	Start point is positioned at the top of the tube.	
At bottom	Start point is positioned at the bottom of the tube.	

Option	Description	Example
At longest segment	Start point is positioned at the longest segment of the tube.	
At smallest segment	Start point is positioned at the smallest segment of the tube.	

Electric box tab

Use the **Electric box** tab to control the shape, number and dimensions of the electric boxes and to define a possible connection piece (**Tube a**).

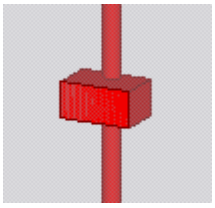
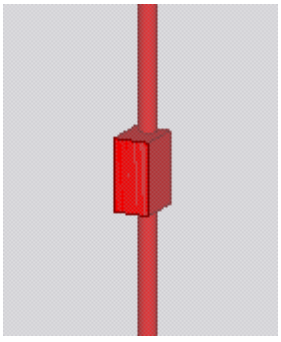
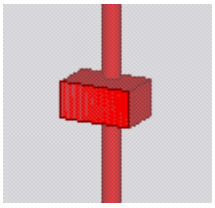
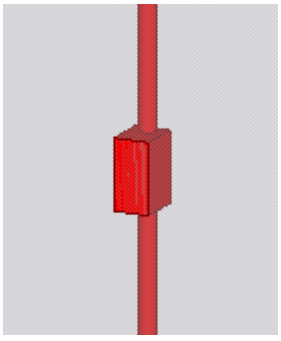
Electric box / Tube a

Option	Description	Default
t, **b**, **h**	Electric box and the connection piece (**Tube a**) thickness, width and height.	`D19`
Pos_No	Prefix and a start number for the part position number.	`ET 1`
Material	Material grade.	`PVC`
Name	Name for the electric box and the connection piece.	`E-tube`

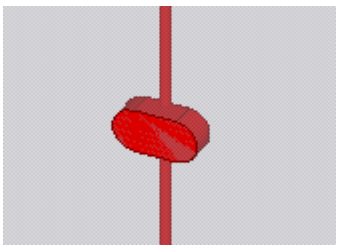
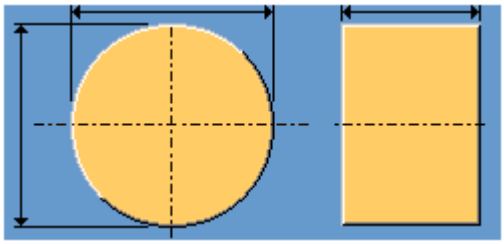
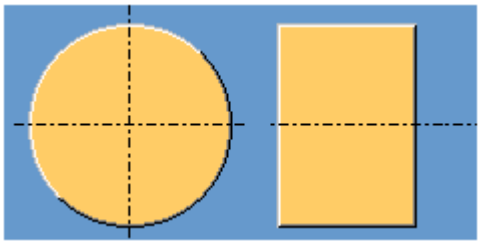
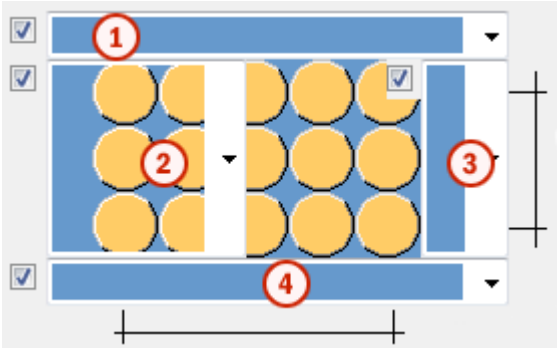
Option	Description	Default
Class	Part class number for the electric box and the connection piece.	2
Comment	Add a comment for the electric box and the connection piece.	

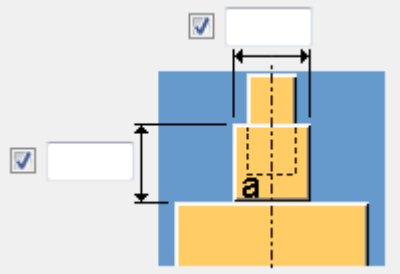
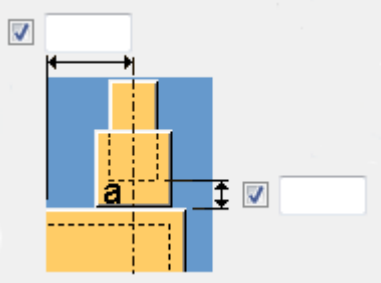
Rotation

Control the rotation of the electric box and the connection piece.

Option	Example
Front	
Top	
Back	
Below	

Electric box shape

Option	Description
Profiles / **Custom part** Example custom part: 	You can use a profile from the profile catalog or select a custom component from the **Applications & components** catalog. If you use a custom component, browse for the component and use the list of options to position the custom component. To use saved custom component properties, select the saved properties file.
	Select the shape of the electric box and define the dimensions for the electric box.
	Select an extra shape for creating several different types of electric boxes. This shape is visible in drawings and it indicates the front and back side of the electric box.
Number =	Number of electric boxes in horizontal or vertical direction.
	
1	Select the top connection pieces for the electric box.
2	Select the left side connection pieces for the electric box.

Option	Description
3	Select the right side connection pieces for the electric box.
4	Select the bottom connection pieces for the electric box.
	Dimensions of the connection pieces.
	Positions of the connection pieces.
	Options to add the electric box to the part. • Create electric box • Create electric box + cut electric box • Create electric box + cut bounding box

Top conn / Bottom conn tab

Use the **Top conn** and **Bottom conn** tabs to control the shape and dimensions of the top and bottom connection boxes, and to define a possible connection piece (**Tube a**).

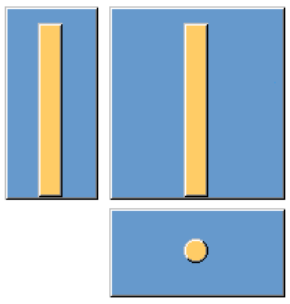
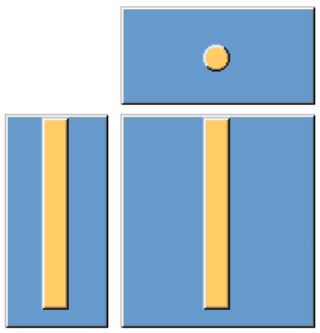
Top box / Bottom box / Tube a

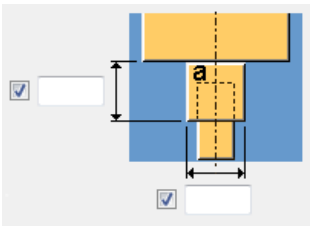
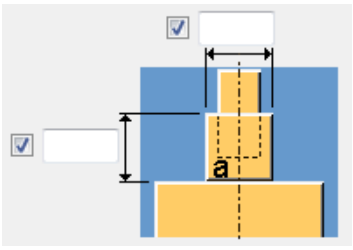
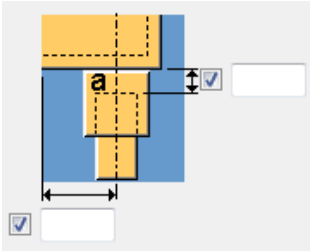
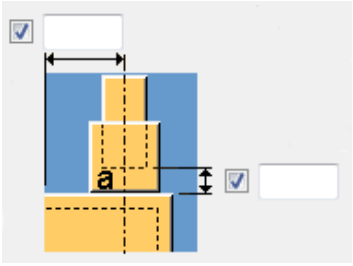
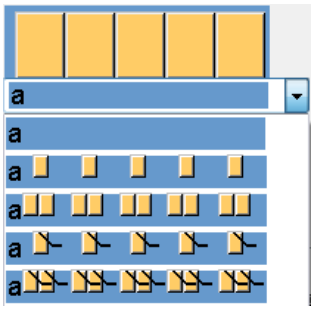
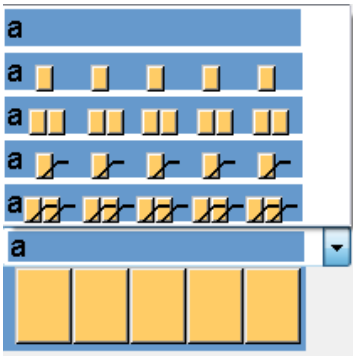
Option	Description
t, **b**, **h**	Top and bottom connection box and the connection piece (**Tube a**) thickness, width and height.
Pos_No	Prefix and a start number for the part position number.
Material	Material grade.
Name	Name for the connection box and the connection piece.
Class	Part class number for the connection box and the connection piece.
Comment	Add a comment for the connection box and the connection piece.

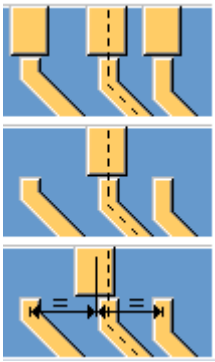
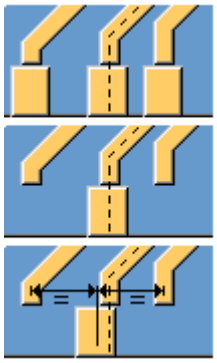
Rotation

Control the rotation of the connection piece (**Tube a**).

Connection box shape

Top conn	Bottom conn	Description
Profiles / **Custom part**		You can use a profile from the profile catalog or select a custom component from the **Applications & components** catalog. If you use a custom component, browse for the component and use the list of options to position the custom component.
		Select the shape of the connection box and define the dimensions for the box.

Top conn	Bottom conn	Description
		Dimensions of the connection box.
		Position of the connection piece.
		Options to add the connection box to the part. • Create connection box • Create connection box + cut connection box • Create connection box + cut bounding box
		Select the connection pieces for the connection box.

Top conn	Bottom conn	Description
		Select the number of connection boxes.

Connect

Define how to the connection box is connected.

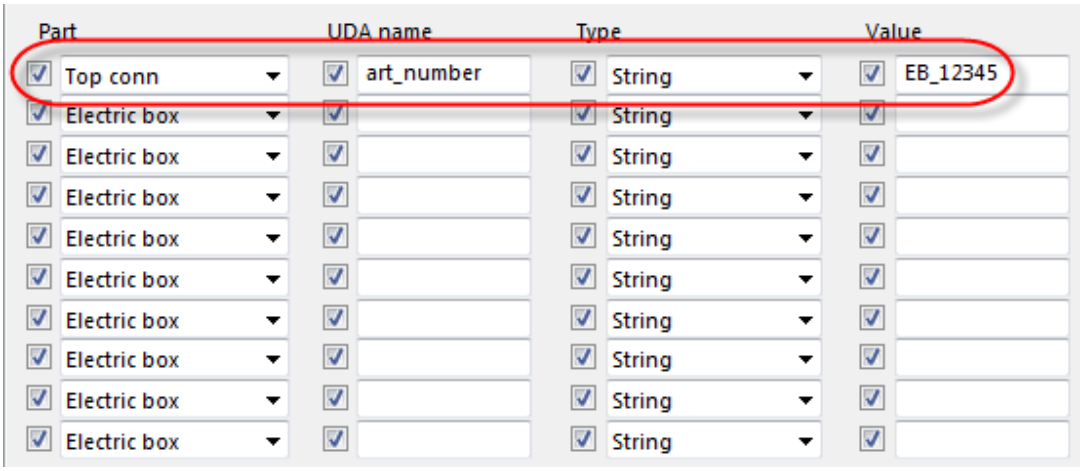
UDA tab

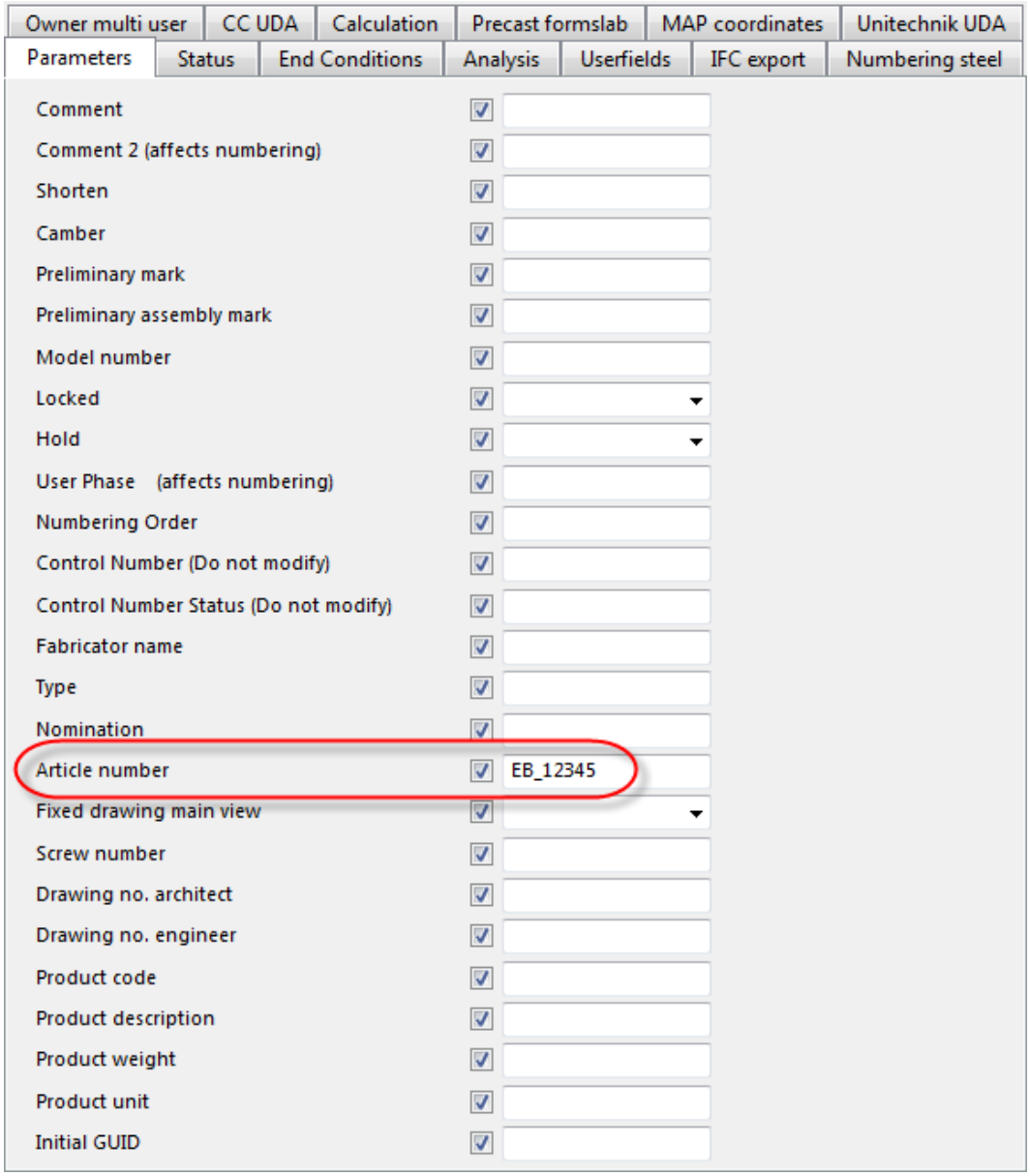
Use the **UDA** tab to add information in the parts' user-defined attributes (UDAs).

Option	Description		
Part	Select to which part the related information can be saved: **Nothing** **Tube top** **Tube bottom** **Tubes** **Electric box** **Top connection** **Bottom connection** **All**		
UDA name	Enter the name of the user-defined attribute. For example, to add a comment UDA, open the `objects.inp` file in a text editor and search for `comment`. The following attribute is shown: `attribute("comment", "j_comment", string,`	**UDA name:** • comment • fabricator • art_number • type	**UDA:** • Comment • Fabricator name • Article number • Type

Option	Description		
	`"%s", no, none, "0.0", "0.0")` The first text between the quotation marks is the UDA name, `comment`. The entered name is case sensitive.		
Type	Type of the user-defined attribute.	Use **String** for text, **Integer** for numbers, **Float** for numbers with decimals and **Option** for selecting an item in a list. You can find the UDA type in the `objects.inp` file	
Value	Enter the value that is saved to the user-defined attribute.		

Example





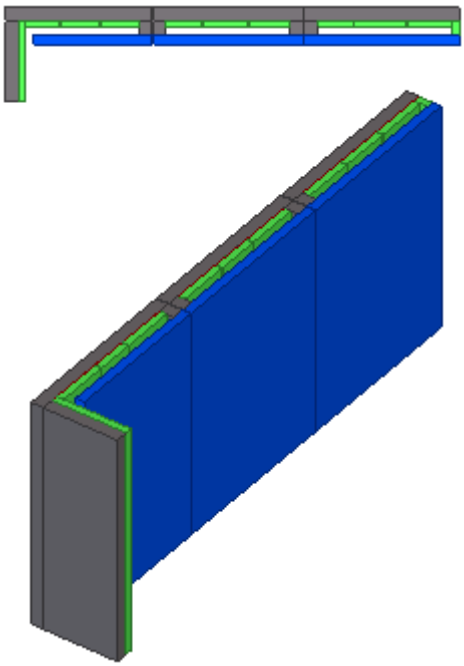
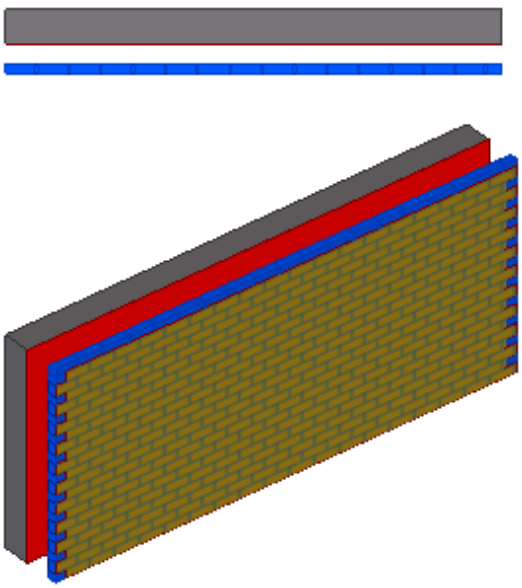
Sandwich and double wall

Sandwich and double wall creates a precast concrete wall. The wall consists of an inner shell, foil, insulation, and outer shell.

Objects created

- Inner shell
- Foil (optional)
- Insulation (optional)
- Outer shell (optional)

Use for

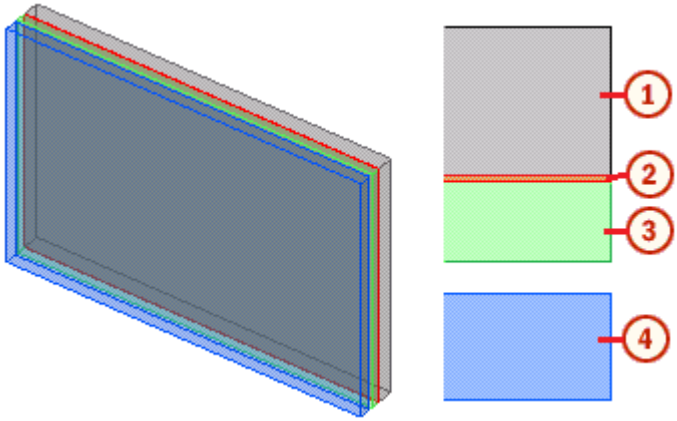
Situation	Description
	Sandwich wall, split shells.
	Sandwich wall, brick wall on outer shell as surface treatment.

Selection order

1. Pick the first point.
2. Pick the second point.

   The wall is created automatically when the second point is picked.

Part identification key

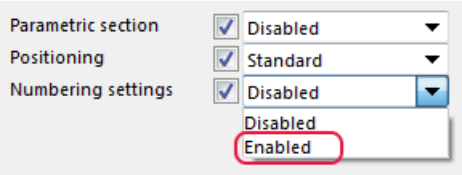


	Part
1	Inner shell
2	Foil
3	Insulation
4	Outer shell

Parts tab
Use the **Parts** tab to control the dimensions and positioning of the sandwich wall inner shell, foil, insulation, and outer shell. The inner shell is always created.

Numbering settings

Set **Numbering settings** to **Enabled** to show **Prefix** and **Start number** for all parts.



Inner shell

Option	Description	Default
Thickness	Define the thickness of the inner shell.	`150 mm`

Foil

Option	Description	Default
Thickness	Define the thickness of the foil.	`1 mm`
Create foil	Define whether the foil is created.	`Yes and subassembly`

Insulation

Option	Description	Default
Thickness	Define the thickness of the insulation.	`50 mm`
Create insulation	Define whether the insulation is created.	`Yes and subassembly`
Edge strips in split case	Define different material and class properties for parts on the edge of the wall.	

Outer shell

Option	Description	Default
Thickness	Define the thickness of the outer shell.	`1 mm`
Create outer shell	Define whether the outer shell is created.	`Yes + cast unit`

Properties

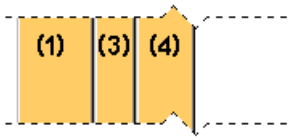
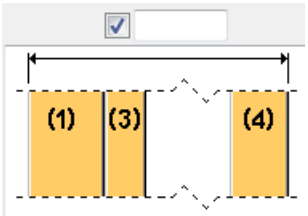
Option	Description	Default
Pos_No	Prefix and start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options** .
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options** .

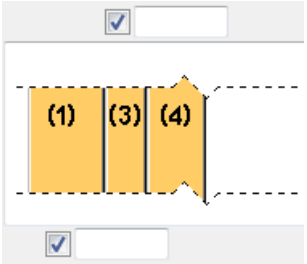
Option	Description	Default
Name	Name that is shown in drawings and reports.	
Class	Part class number.	
Cast unit type	Defines if the structure type of the part is **Precast** or **Cast in place**. Set **Cast unit type** to **Enabled** to show **Cast unit type** for parts.	**Precast**

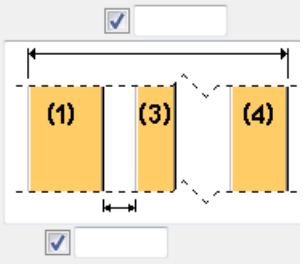
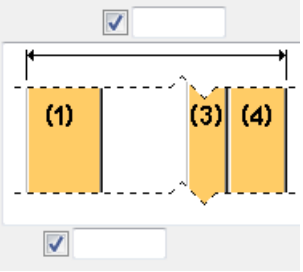
Positioning

You can use **Standard** or **Advanced** positioning. With advanced positioning, there are more options for sandwich wall width and foil position.

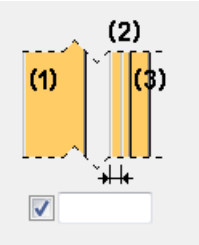
Sandwich wall width

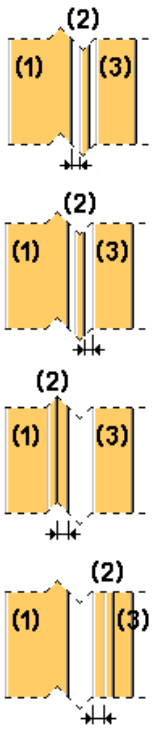
Option: Standard	Description
	No gaps between the parts. Sandwich wall thickness is the sum of all created parts.
	Define the total wall width. A gap is created between the outer shell and the insulation.

Option: Advanced	Description
	Define the total wall width and the gap between the inner shell and the insulation. Another gap is created between the outer shell and the insulation.

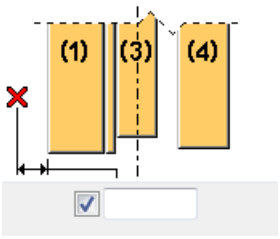
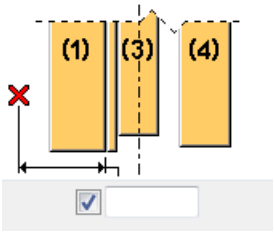
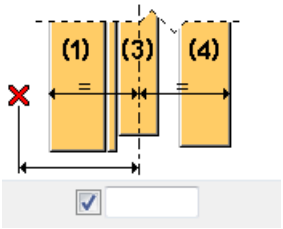
Option: Advanced	Description
	Define the total wall width and the gap between the outer shell and the insulation. Another gap is created between the inner shell and the insulation.
	Define the total wall width and the gap between the inner shell and the insulation. No gap is created between the outer shell and the insulation.

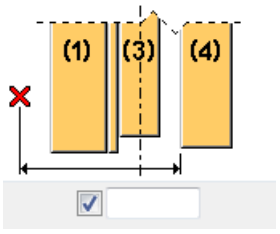
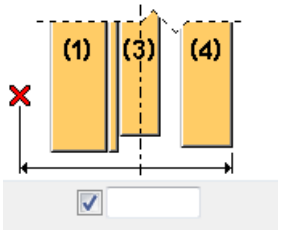
Foil position

Option	Description
	By default, the foil is placed on the outer side of the inner shell.

Option	Description
	Define another foil position. Set **Positioning** to **Advanced**.

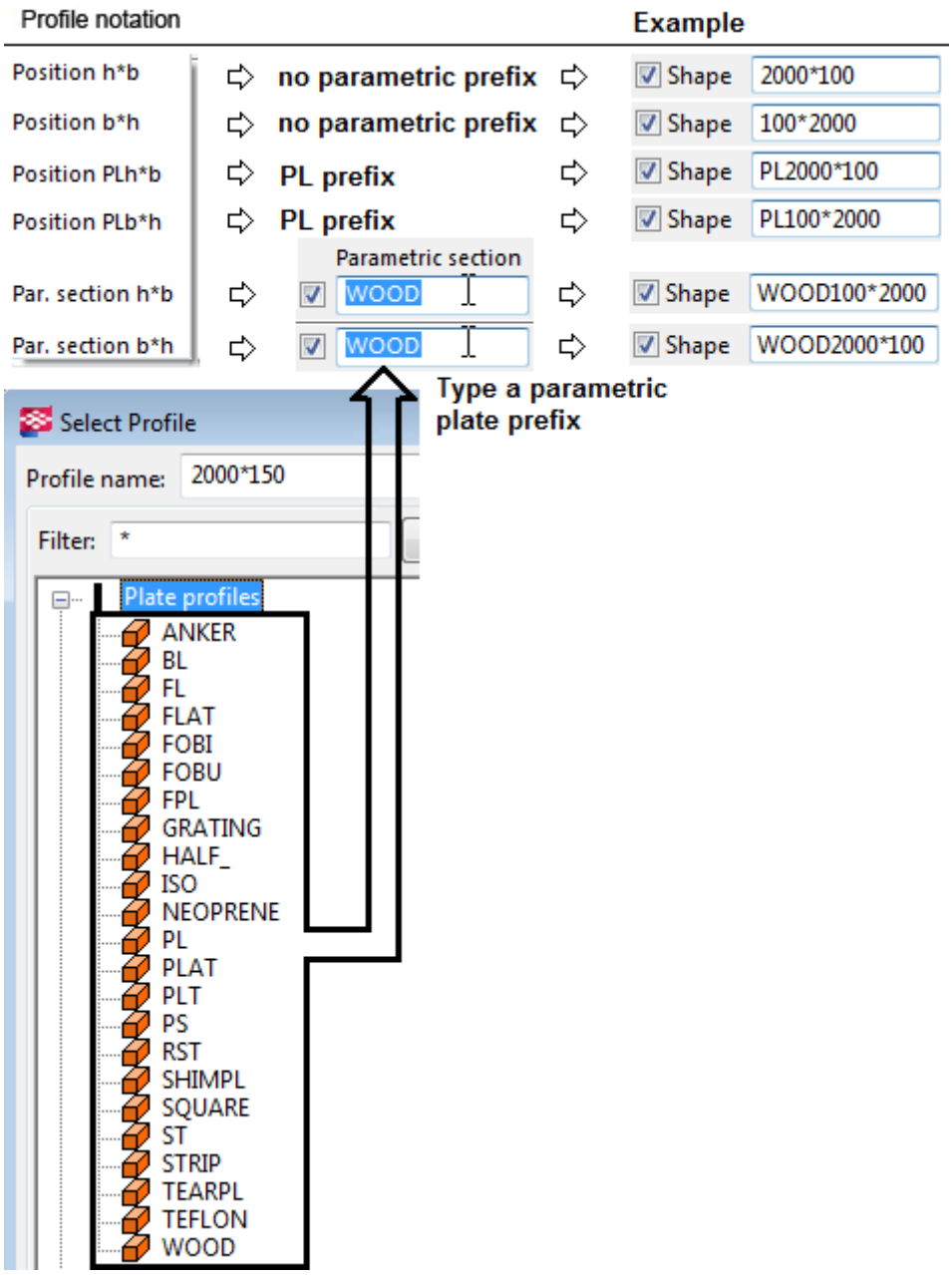
Point offset

Option	Description
	Define the offset from the outer face of the inner shell.
	Define the offset from the inner face of the inner shell.
	Define the offset from the center of the sandwich wall.

Option	Description
	Define the offset from the inner face of outer shell.
	Define the offset from the outer face of outer shell.

Parametric profile

Option	Description
Parametric section	Set to **Enabled** to define the prefix for the parametric profiles of the sandwich parts.
Prefix for the parametric profile	Select the prefix for the parametric profile: • `Position h*b` and `Position b*h` to create plate profiles without a prefix. If **Parametric section** is set to **Disabled**, the parts have an `h*b` prefix, for example, `2000*100`. • `Position PLh*b` and `Position PLb*h` to create plate profiles whose prefix starts with `PL`. • `Par. section h*b` and `Par. section b*h` require you to enter a parametric plate profile prefix.
Par.section prefix	Define the prefix of the parametric profile.



Vertical section tab

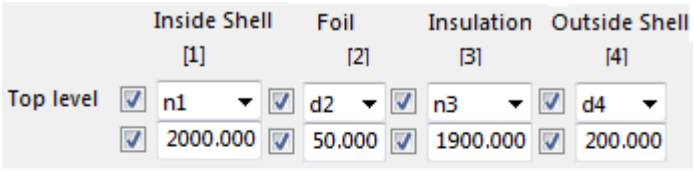
Use the **Vertical section** tab to control the vertical level part properties. You can define both the top and the bottom level. The levels can be set as absolute levels, or vertically relative (displaced) to the points.

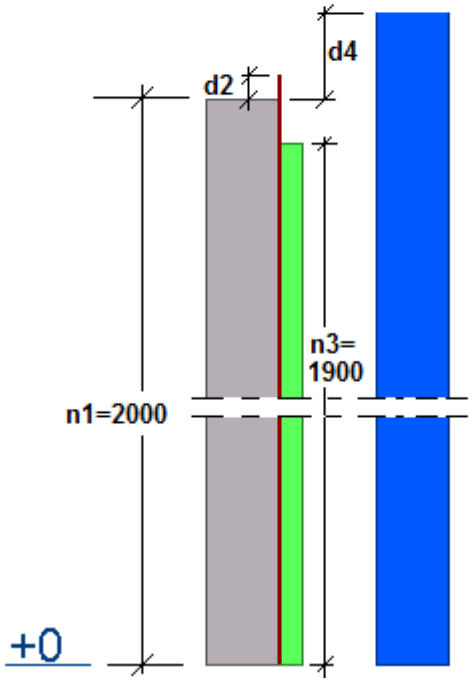
Top level

Option	Description
n1, **n2**, **n3**, **n4**	Define the absolute top level height.

Option	Description
h1, **h2**, **h3**, **h4**	Define the part height from the bottom face.
d2, **d3**, **d4**	Define the vertical displacement from the adjacent part.

When you use relative levels, the levels of all other parts depend on the inner shell level.

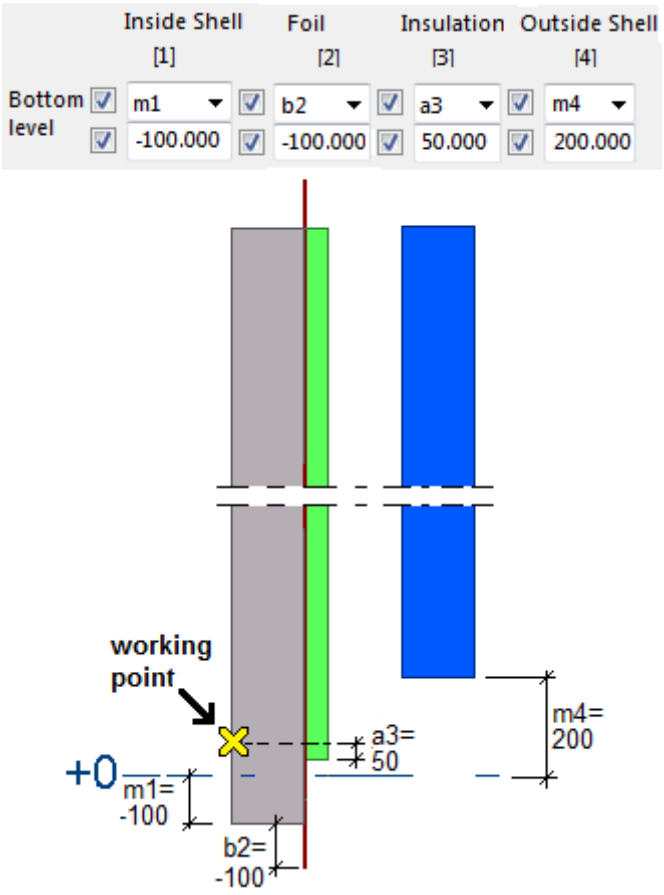




Bottom level

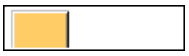
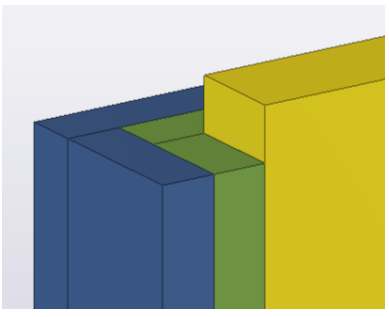
Option	Description
m1, **m2**, **m3**, **m4**	Define the absolute bottom level height.
a1, **a2**, **a3**, **a4**	Define the bottom offset from the point.
b2, **b3**, **b4**	Define the vertical displacement from the adjacent part.

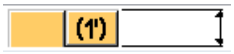
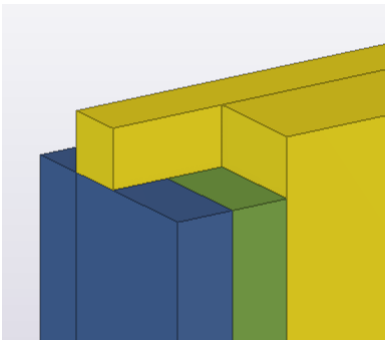
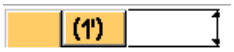
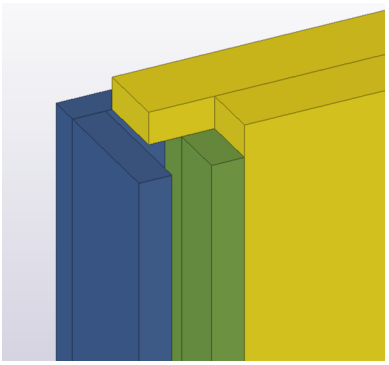
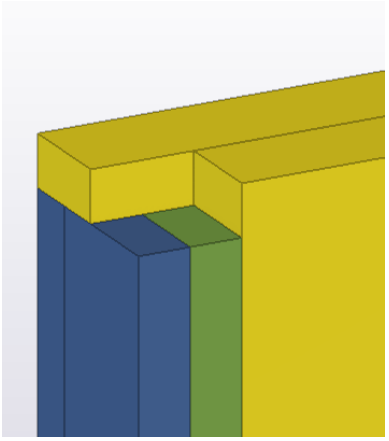
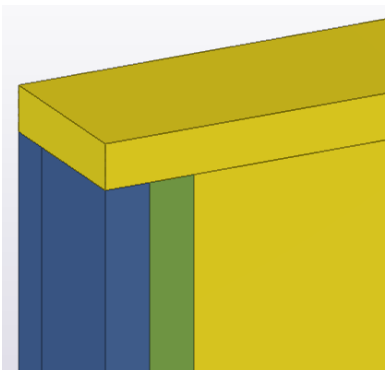
When you use relative levels, the levels of all other parts depend on the inner shell level.

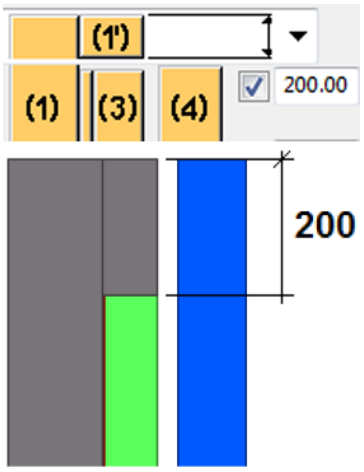
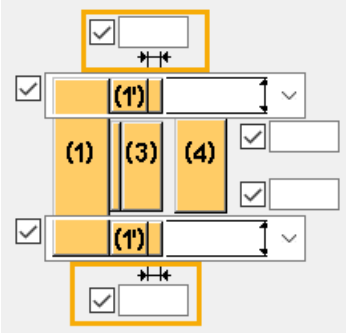
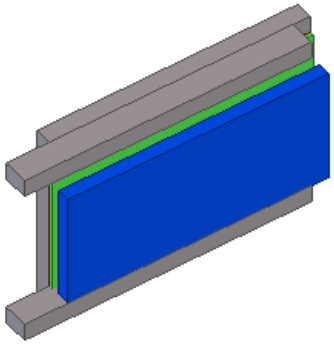


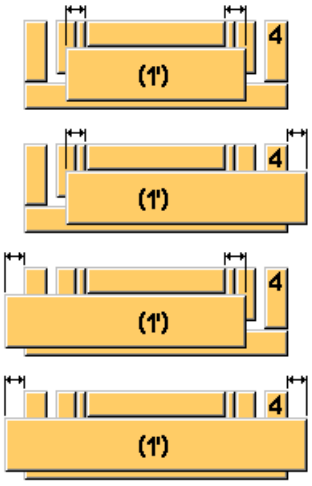
Thicken innershell

You can define a thickening for both the top and the bottom side of the inner shell.

Option	Description	Example
	Inner shell is not thickened.	

Option	Description	Example
	Inner shell is thickened towards the outer face of the insulation. Define the insulation height to prevent collisions.	
	Inner shell is thickened to the inner face of the outer shell.	
	Inner shell is thickened to the outer face of the outer shell. Define the outer shell height to prevent collisions.	
	Inner shell covers the insulation and outer shell.	

Option	Description	Example
Thickness	Define the inner shell thickness. If you do not enter a value, the inner shell thickness defined on the **Parts** tab is used.	
Thickening parts	Select how the top and bottom thickening parts are added to the inner shell.	
Gap in inner shell	Enter a value to create a gap between the inside and the front of the inner shell at the bottom and the top. The gap is filled with insulation. 	
Offset options	Select the offset options for the thickening parts. Offset	

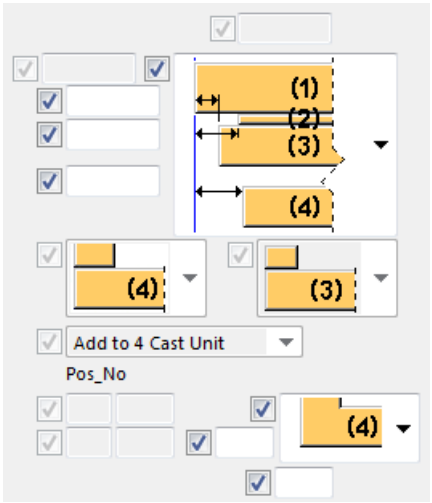
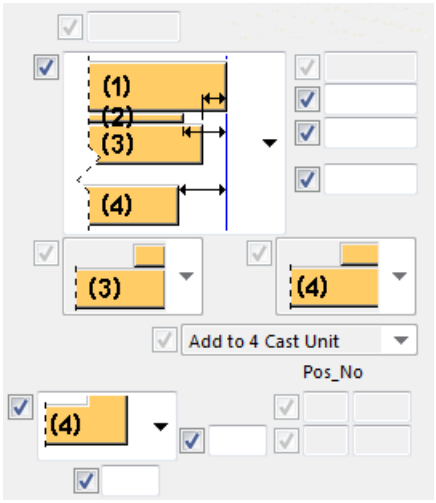
Option	Description	Example
	can be defined for both wall ends. 	

Horizontal section tab

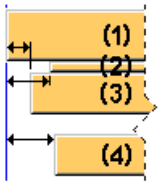
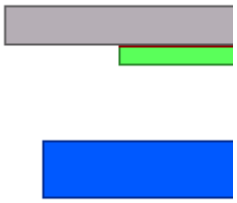
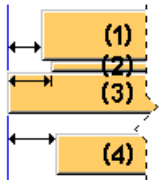
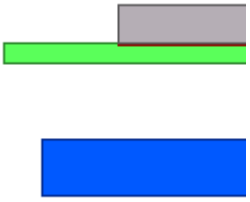
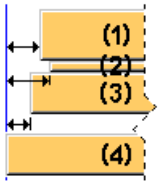
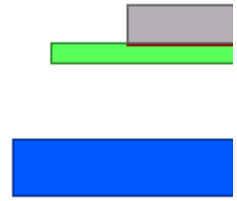
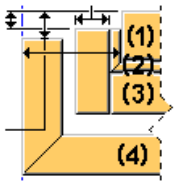
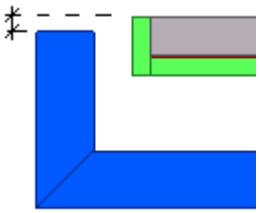
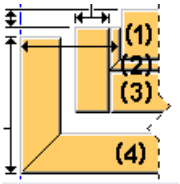
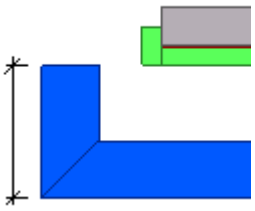
Use the **Horizontal section** tab to define the wall ends and to control how the wall parts are split in the horizontal direction.

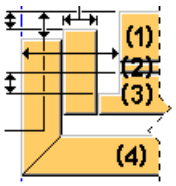
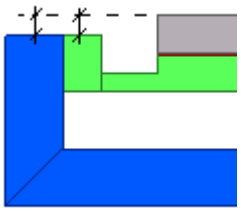
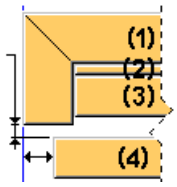
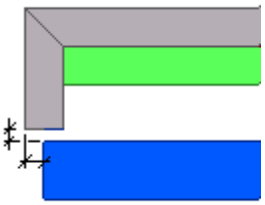
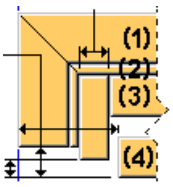
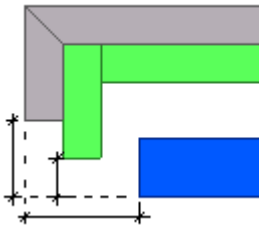
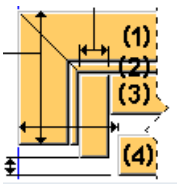
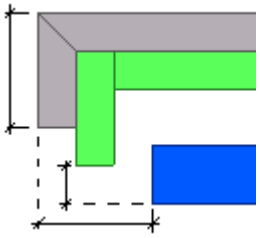
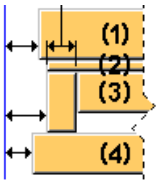
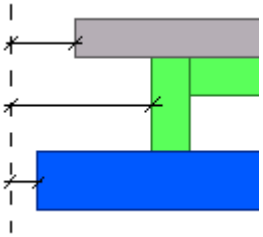
Wall end

The wall end options are the same for both ends.

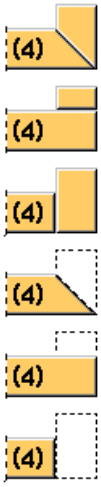
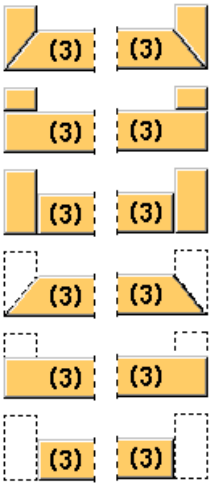
First wall end	Second wall end
	

Wall end offset

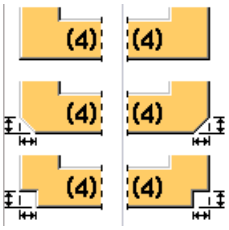
Option	Description	Example
	Inner shell end is fixed. Define the offset for the ends of the foil, insulation, and outer shell.	
	Insulation end is fixed. Define the offset for the ends of the inner shell, foil, and outer shell.	
	Outer shell end is fixed. Define the offset for the ends of the inner shell, foil, and insulation.	
	Corner shape to the inside direction. For the outer shell angle, define the offset from the inner face of the inner shell. Insulation is along the inner shell.	
	Corner shape to the inside direction. Define the length of the outer shell angle.	

Option	Description	Example
	Corner shape to the inside direction. For the outer shell angle, define the offset from the inner face of the inner shell. Insulation is along the outer shell.	
	Corner shape to the outside direction. Define the horizontal offset for the outer shell. There is no gap between the insulation and the inner shell angle. The empty void between the front of the created corner part and the front of the insulation is by default filled with insulation.	
	Corner shape to the outside direction. Define the horizontal offset for the outer shell. For the inner shell angle, define the offset from the outer face of the inner shell.	
	Corner shape to the outside direction. Define the horizontal offset for the outer shell. For the inner shell angle, define the length.	
	Angled shells are not created. Define horizontal offset for the inner shell, insulation, and outer shell.	

Corner shape

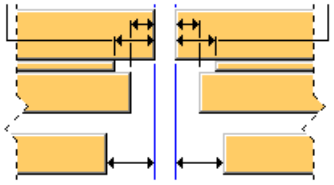
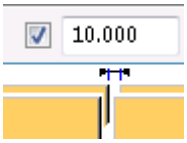
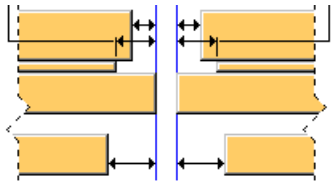
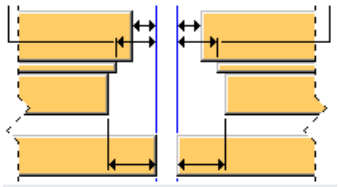
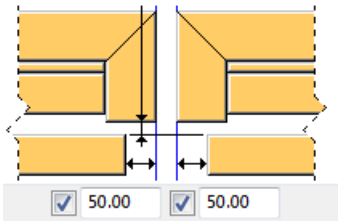
Option	Description
	Select the inner shell corner shape.
	Select the outer shell corner shape.
	Select the insulation corner shape.
Add corner to main shell	Add the corner to the shell. The options are: • `Add to 4 Cast Unit` (4 = outer shell) • `Add to 4 Sub assembly` • `Add to 1 Cast Unit` (1 = inner shell) • `Add to 1 Sub assembly`

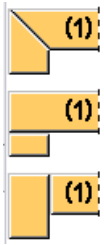
Vertical chamfer

Option	Description
	Define whether a vertical chamfer is created. Define the horizontal and vertical chamfer dimensions, and the part prefix and the start number.

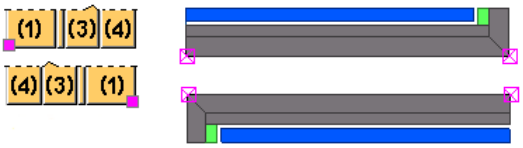
Horizontal offset

To use the options, set the **Split Front view** option to **Enabled**.

Option	Description
	Define the horizontal offset for the foil, insulation, and outer shell. The main gap between the inner shells is defined in the **Split Front view** section of the tab. 
	Define the horizontal offset for the inner shell, foil, and outer shell. The main gap between the insulation parts is defined in the **Split Front view** section of the tab.
	Define the horizontal offset for the inner shell, foil, and insulation. The main gap between outer shells is defined in the **Split Front view** section of the tab.
	Define a 90-degree angle for the inner shell and the vertical offset towards the angled shell from the inner face of the outer shell. The main gap between the inner shells is defined in the **Split Front view** section of the tab. Additional options for the 90-degree angle:

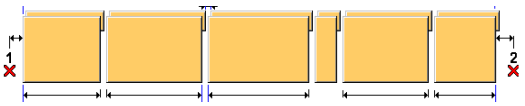
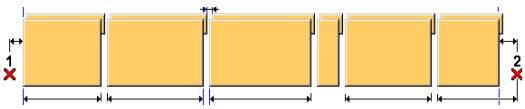
Option	Description
	• Define the corner shapes for the inner shell.  • Define how the corner is added to the inner shell. • Define the part prefix and the start number.

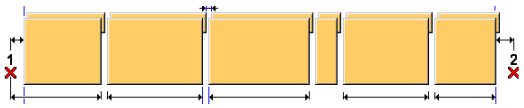
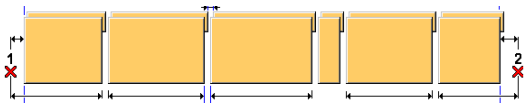
Mirroring

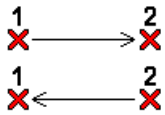
Option	Description
 	Define whether the sandwich wall is mirrored.

Splitting parts

To split wall parts, set the **Split Front view** option to **Enabled**.

Option	Description
 	Parts are not split. Define the horizontal offset for both wall ends using the points as reference points.
 	Parts are split. Define the section width and the number of sections. First end offset is the reference for the first section width at the first wall end. Second end offset is the reference for the last section width at the second wall end.
 	Parts are split. Define the section width and the number of sections. First end offset is the reference for the first section width at the first wall end. Point (2) is the reference of the last section width at the second wall end.

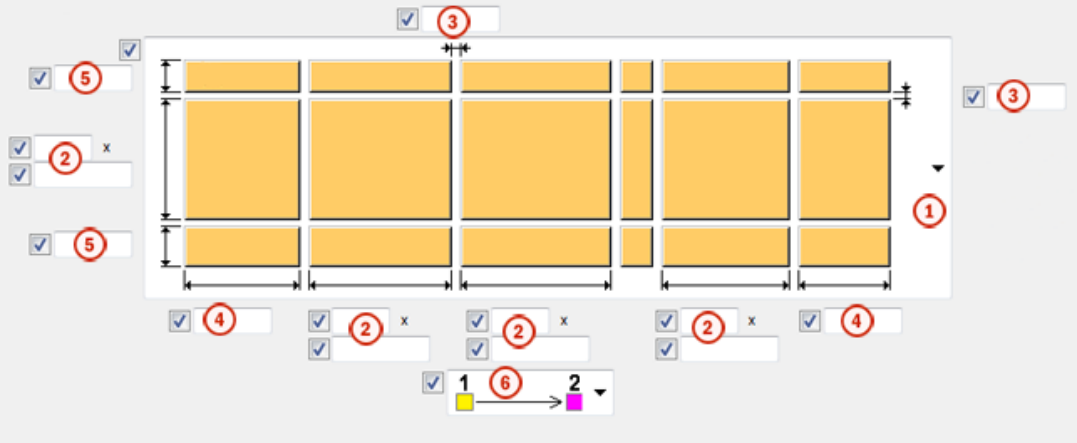
Option	Description
	Parts are split. Define the section width and enter the number of sections. Point (1) is the reference of the start section at the first wall end. Second end offset is the reference for the last section width at the second wall end.
	Parts are split. Define the section width and enter the number of sections. Points (1, 2) are the references for the section width at both wall ends.

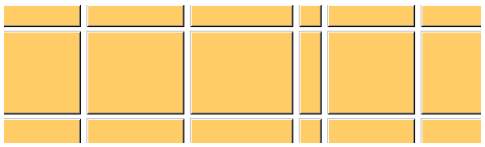
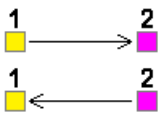
Option	Description
	Define the horizontal direction.

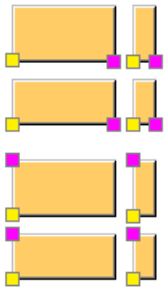
Foil tab

Use the **Foil** tab to control the vertical and/or horizontal splitting of the foil.

Dimensions

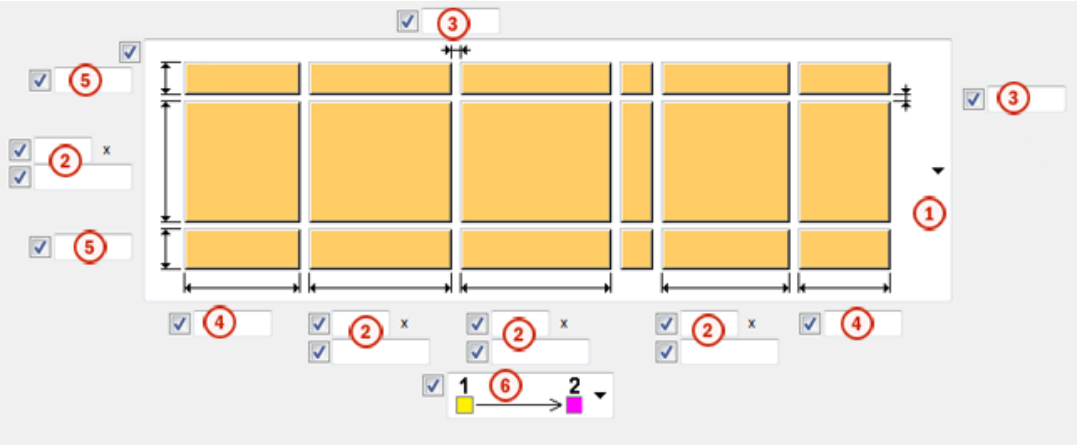


Option	Description
1	Define whether the foil is split: • Foil is not split. • Foil is split in the horizontal direction. • Foil is split in the vertical direction. • Foil is split in the horizontal and vertical direction. 
2	Define the number and width of the middle sections.
3	Define the gap between the sections.
4	Define the width of the first and last section in the horizontal direction.
5	Define the height of the top and bottom section in the vertical direction.
6	Define which wall end is used for distributing the insulation sections. 

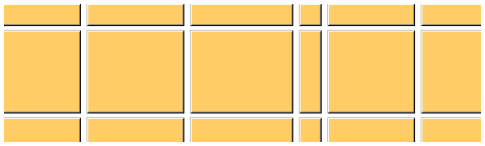
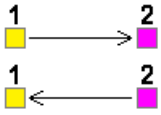
Option	Description
Point direction	Define the direction of the points. The options are horizontal or vertical.  This option affects the representation of workshop drawings.

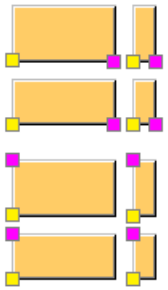
Insulation tab
Use the **Insulation** tab to control the vertical and/or horizontal splitting of the insulation.

Dimensions



Option	Description
1	Define whether the insulation is split: • Insulation is not split.

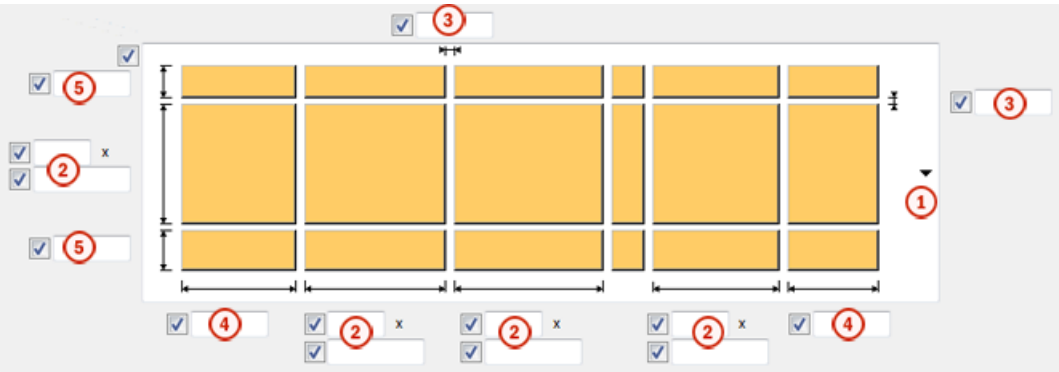
Option	Description
	• Insulation is split in the horizontal direction. • Insulation is split in the vertical direction. • Insulation is split in the horizontal and vertical direction. 
2	Define the number and width of the middle sections.
3	Define the gap between the sections.
4	Define the width of the first and last section in the horizontal direction.
5	Define the height of the top and bottom section in the vertical direction.
6	Define which wall end is used for distributing the insulation sections. 

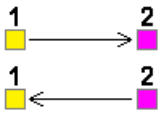
Option	Description
Point direction	Define the direction of the points. The options are horizontal or vertical.  This option affects the representation of workshop drawings.

Outside Shell tab

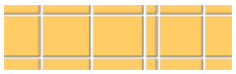
Use the **Outside shell** tab to control the vertical and/or horizontal splitting of the outer shell. You can also define chamfers and add a brick wall surface.

Dimensions

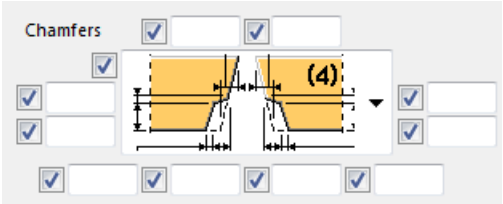


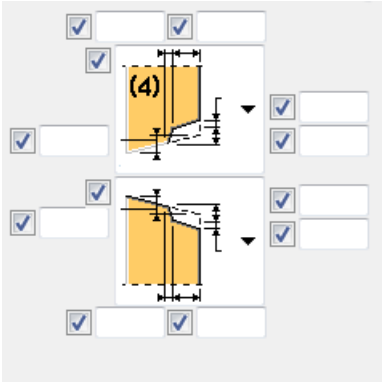
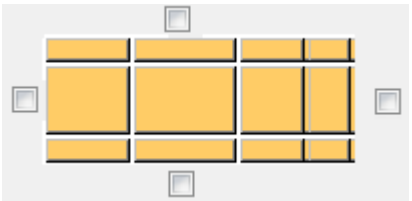
Option	Description
	Define the direction for the splitting.

	Description
1	Define whether the outer shell is split: • Outer shell is not split.

	Description
	• Outer shell is split in the horizontal direction. • Outer shell is split in the vertical direction. • Outer shell is split in the horizontal and vertical direction. 
2	Define the number and width of the middle sections.
3	Define the gap between the sections.
4	Define the width of the first and last section in the horizontal direction.
5	Define the height of the top and bottom section in the vertical direction.

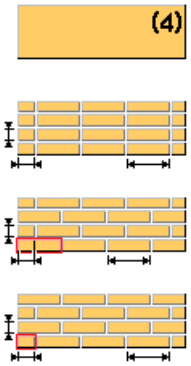
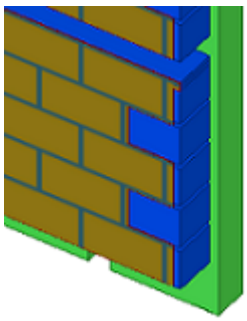
Chamfering

Option	Description
Vertical chamfers	Define the vertical chamfers. Select a chamfer shape from the list and define the chamfer dimensions. 

Option	Description
Horizontal chamfers	Define the horizontal chamfers. Select a chamfer shape from the list and define the chamfer dimensions. 
Chamfer side	Define the sides where the chamfers are created.  For example:

Brick wall properties

Set the **Brick wall** option to **Enabled** to create a brick wall.

Option	Description	Example
	Select the brick wall type for the outer shell and define the brick dimensions, mortar height and width.	
Wall edges	Define whether the wall edges are created as loose parts or whether they are added into an assembly.	

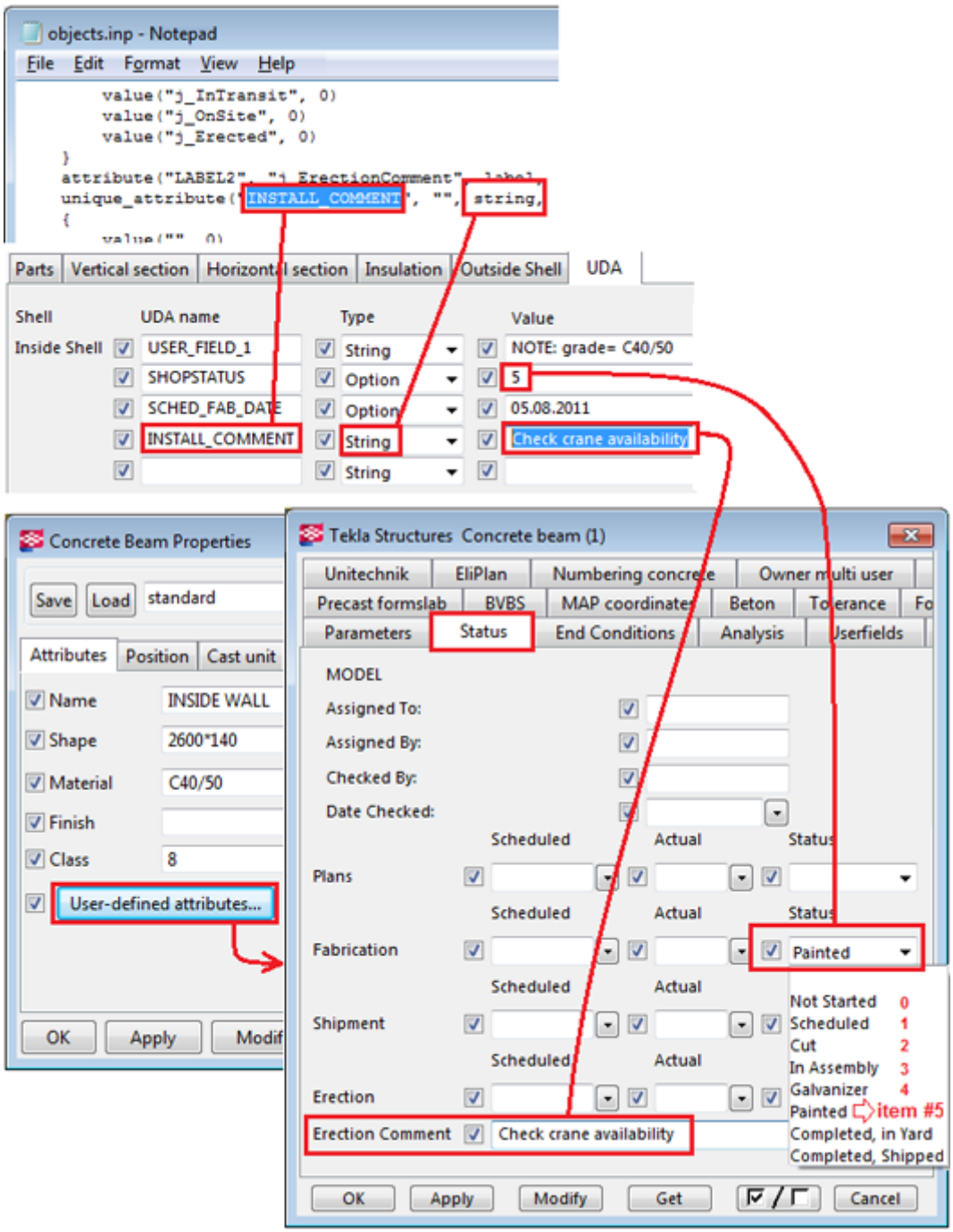
UDA tab

Use the **UDA** tab to define user-defined attributes for the inner shell, foil, insulation, and outer shell. You can define multiple UDAs for each part. UDAs can be used as filters, and they can be displayed in drawings and reports.

Option	Description		
Shell	Select the shell for the user-defined attribute (UDA) that you are defining. You can set more than one UDA for a certain shell, if needed.		
UDA name	Enter the name of the user-defined attribute. For example, to add a comment UDA, open the `objects.inp` file in a text editor and search for `comment`. The following attribute is shown: `attribute("comment", "j_comment", string, "%s", no, none, "0.0", "0.0")` The first text between the quotation marks is the UDA name, `comment`. The entered name is case sensitive.	**UDA name:** • comment • fabricator • art_number • type	**UDA:** • Comment • Fabricator name • Article number • Type

Option	Description	
Type	Type of the user-defined attribute.	Use **String** for text, **Integer** for numbers, **Float** for numbers with decimals and **Option** for selecting an item in a list. You can find the UDA type in the `objects.inp` file
Value	Enter the value that is saved to the user-defined attribute.	

For example:



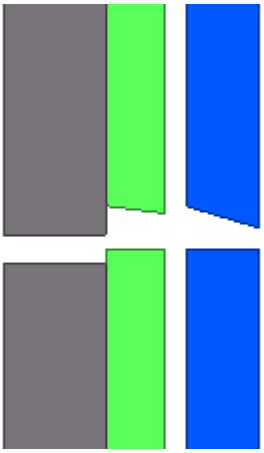
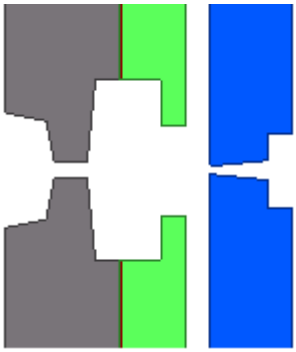
Sandwich wall horizontal seam

Sandwich wall horizontal seam creates a horizontal seam between two sandwich walls. Seam dimensions and rabbets can be defined for all layers: inner shell, foil, insulation, and outer shell. In addition, you can define an extra foil layer.

Objects created

- Seams
- Rabbets
- Foil
- Insulation
- Extra foil layer

Use for

Situation	Description
	Seams between sandwich wall shells.
	Seams with additional rabbets.

Before you start

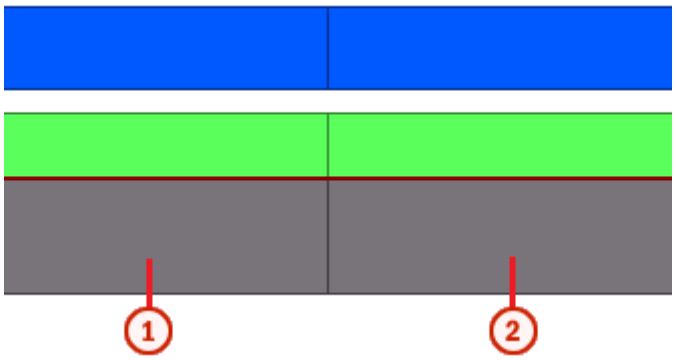
To be able to select the needed parts, activate the **Select objects in components** switch.

Selection order

1. Select the inner shell of the first sandwich wall.
2. Select the inner shell of the second sandwich wall.

   The seam is created automatically when the secondary part is selected.

Part identification key

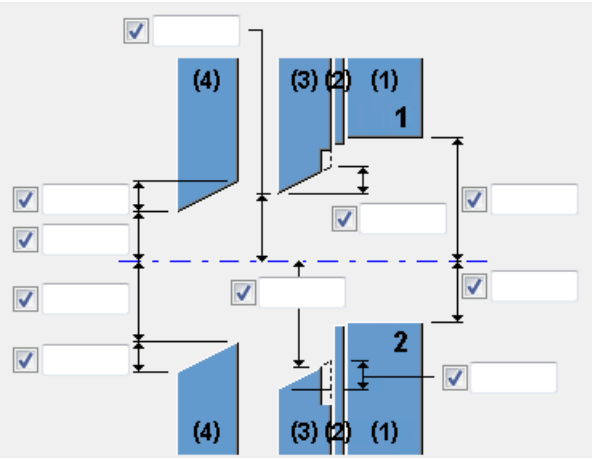


	Part
1	Concrete part (wall, column, beam, slab)
2	Point Multiple points can be picked.

Picture tab
Use the **Picture** tab to control the horizontal seam properties.

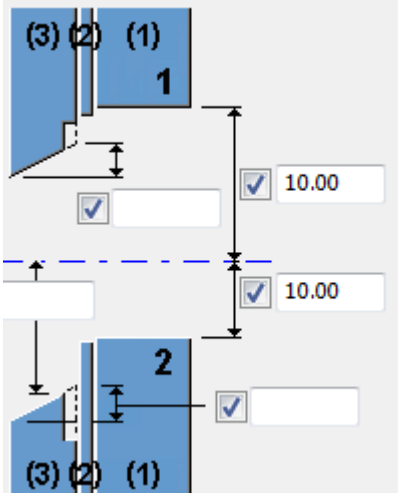
Seam position

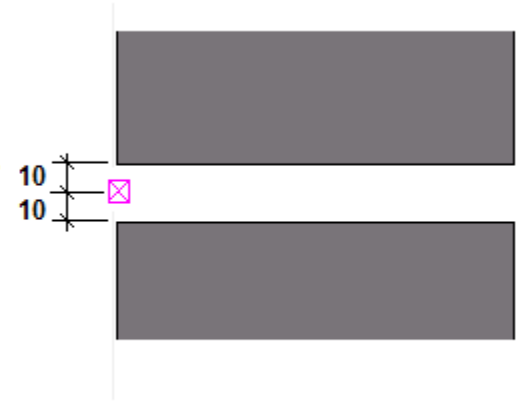
Seams can be defined for inner shells, insulation and outer shells. Draft angles can be defined for the outer shells.



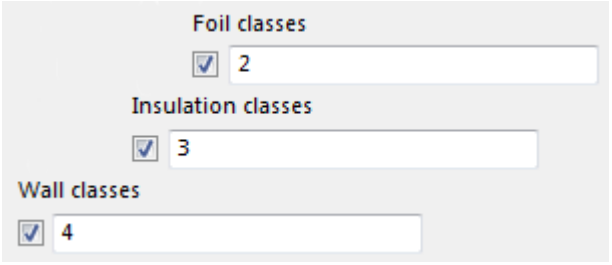
Define the seam position between the sandwich wall shells. The middle line between the walls is used as a reference when the seam position is calculated.

Example:





Shell classes

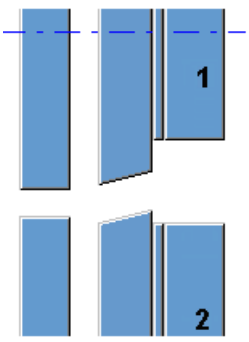


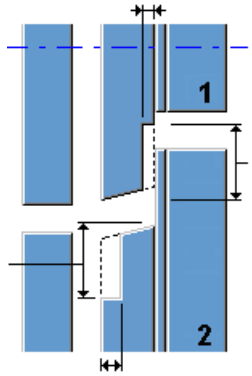
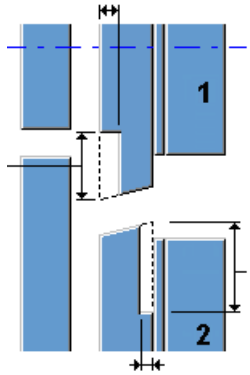
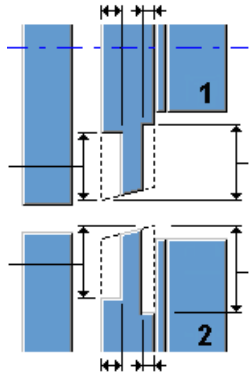
Define the part classes to be applied to the seams. By default, the **Sandwich Wall Horizontal Seam** component creates inner shells with class 1, foil with class 2, insulation with class 3, and outer shells with class 4. Use these classes to apply seams.

To prevent the creation of seams between foils, insulation shells and outer shells, leave the **Insulation classes** box empty.

Rabbets in insulation

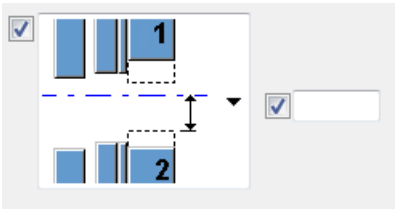
You can create rabbets in the insulation layer.

Option	Description
	No rabbet for the insulation layer.

Option	Description
	One side rabbet for the top and bottom of the insulation layer. Rabbet depth and width can be defined.
	Mirrored one side rabbet for the top and bottom of the insulation layer. Rabbet depth and width can be defined.
	Two sided rabbets for the top and bottom of the insulation layer. Rabbet depth and width can be defined.

Offset

You can define offset for the seam.



Rabbets tab

Use the **Rabbets** tab to define rabbets in inner shells, insulation layers and outer shells.

Rabbet side

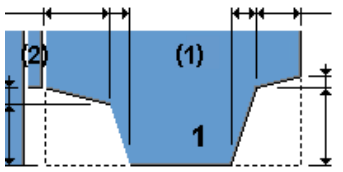
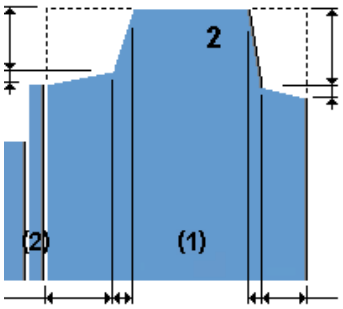
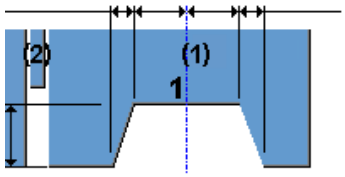
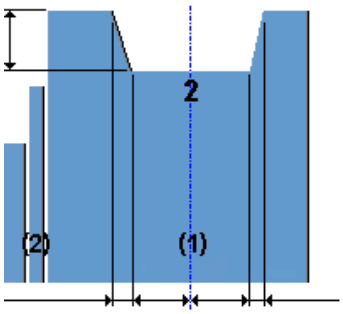
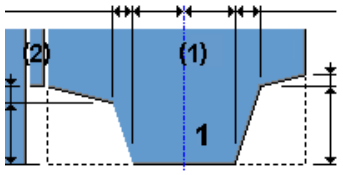
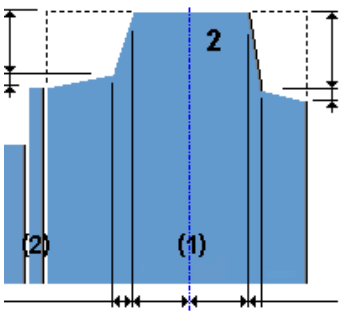
Select to which side of the sandwich wall the rabbets are applied. You can define the rabbets separately for both sides.

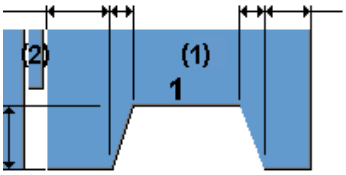
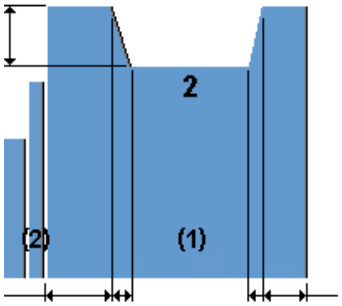
The options are:

- **Top side**
- **Bottom side**
- **Both sides**

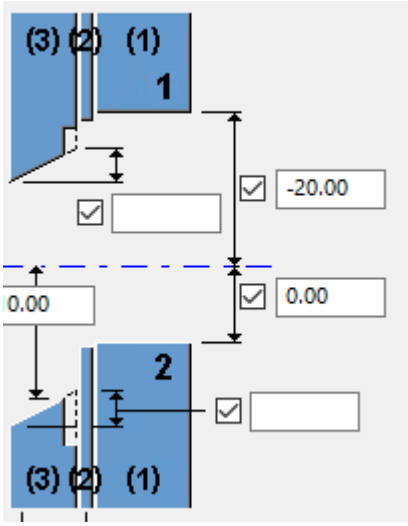
Connection type

You can select a tongue or groove joint connection for both sides.

Option	Option	Description
		Tongue joint connection, based on the outer geometry
		Groove joint connection, based on the center line
		Tongue joint connection, based on the center line

Option	Option	Description
		Groove joint connection, based on the outer geometry

If you create one tongue and one groove joint, it is possible to overlap the parts. This is defined on the **Picture** tab with seam position values. Enter negative values to move the parts closer together.



Extra Foils tab

Use the **Extra foils** tab to define an extra foil layer in the seam.

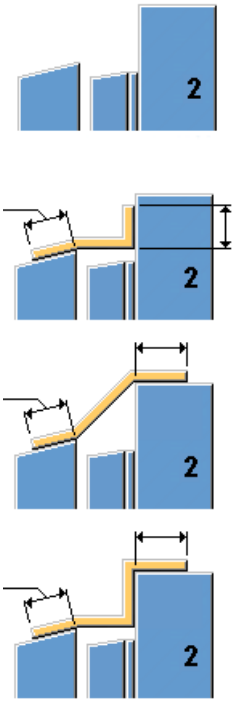
Foil

Option	Description	Default
t	Define the extra foil thickness.	`1 mm`
Pos_No	Prefix and start number for the part position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options** .
Material	Material grade.	The default material is defined in the **Part material** box in the

Option	Description	Default
		Components settings in **File menu** --> **Settings** --> **Options** .
Name	Name for the extra foil.	`FOIL`
Class	Part class number for the extra foil layer.	
Comment	Add a comment for the extra foil layer.	

Connection method

Option	Description
No connection	Select how the extra foil is attached to the sandwich wall. • **No connection** (foil is a loose part) • **Weld** • **Cast unit** • **As subassembly**
At Inside shell	Select the profile to which the extra foil is connected. • **At inside shell** • **At outside shell**
Position h*b	Select a prefix for the foil layer. • **Position h*b** and **Position b*h** create a profile without a prefix. • **Position PLh*b** and **Position PLb*H** create a profile which start with **PL** prefix. • **Par. section h*b** and **Par. section b*h**: enter a prefix in the box below.
Auto	Select the rotation of the foil layer. • **Auto** • **Below** • **Front** • **Top** • **Back**

Option	Description
Foil shape	Define shape of the extra foil. 

General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Sandwich wall vertical seam

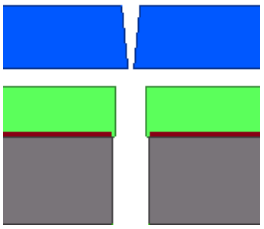
Sandwich wall vertical seam creates a vertical seam between two sandwich walls. Seam dimensions and rabbets can be defined for all layers: inner shell, foil, insulation, and outer shell.

Objects created

- Seam
- Rabbets
- Foil

- Insulation

Use for

Situation	Description
	Seams between sandwich wall shells.
	Seams with additional rabbets.

Limitations

The component works only if the sandwich walls are parallel.

Before you start

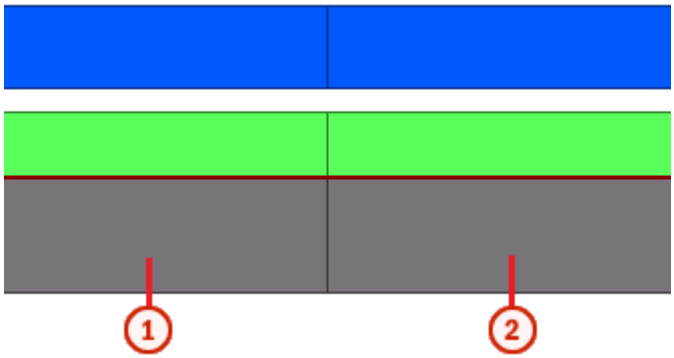
To be able to select the needed parts, activate the **Select objects in components** switch.

Selection order

1. Select the inner shell of the first sandwich wall.
2. Select the inner shell of the second sandwich wall.

   The seam is created automatically when the secondary part is selected.

Part identification key

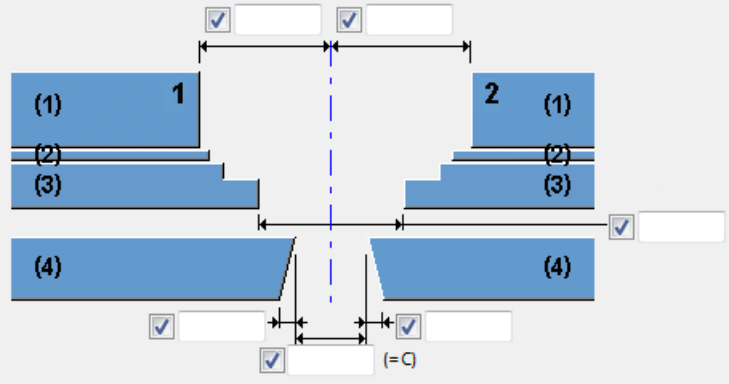


	Part
1	Inner shell of the first sandwich wall
2	Inner shell of the second sandwich wall

Picture tab
Use the **Picture** tab to control the vertical seam properties.

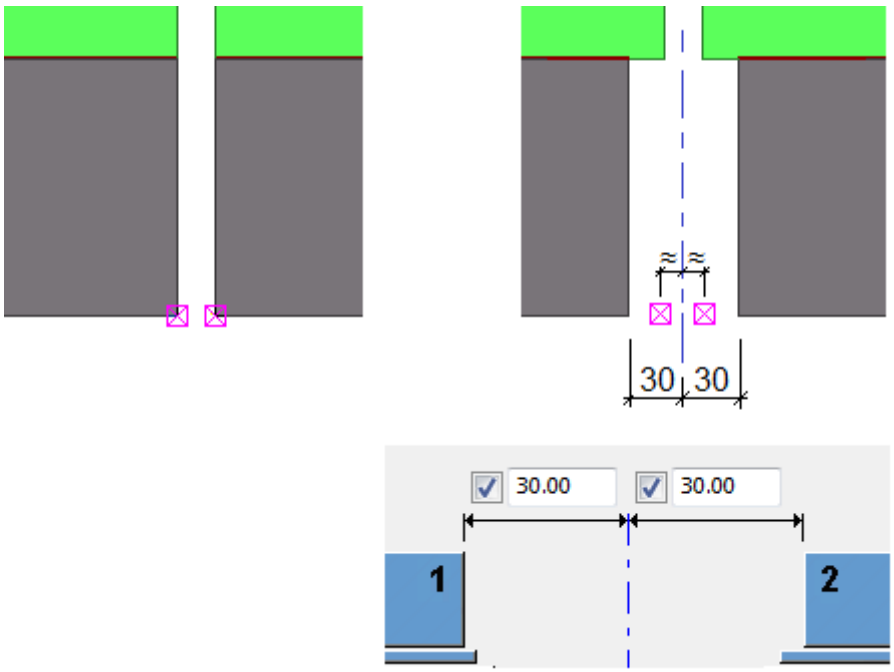
Seam position

Seams can be defined for inner shells, insulation and outer shells. Draft angles can be defined for the outer shells.

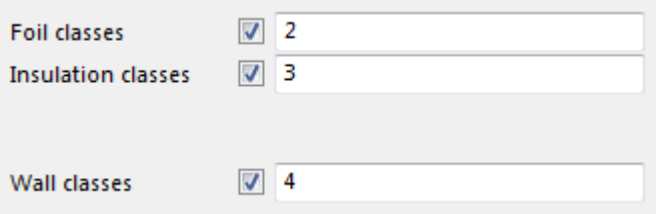


Define the seam position between the sandwich wall shells. The middle line between the walls is used as a reference when the seam position is calculated.

Example:



Shell classes

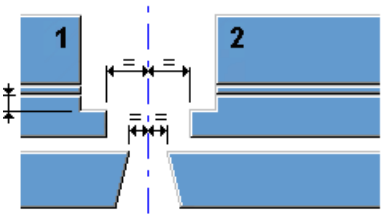
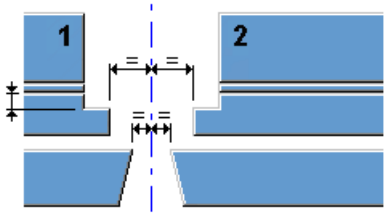
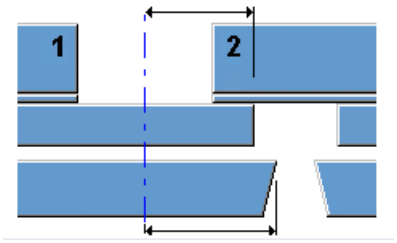
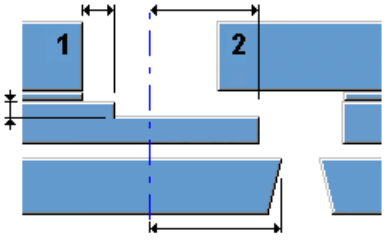


Define the part classes to be applied to the seams. By default, the **Sandwich Wall Vertical Seam** component creates inner shells with class 1, foil with class 2, insulation with class 3 and outer shells with class 4. Use these classes to apply seams.

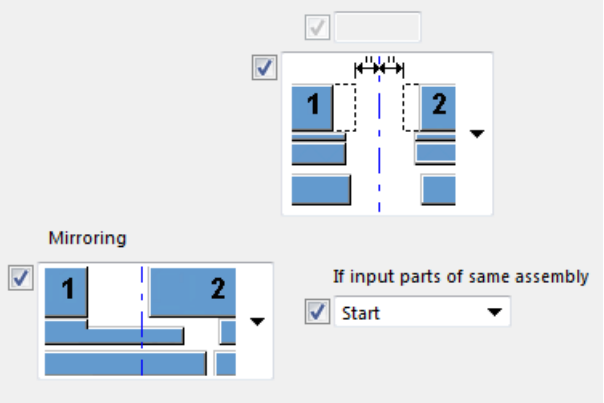
For example, to prevent the creation of seams between foils, insulation shells and outer shells, leave the **Insulation classes** box empty.

Asymmetrical seams

You can create asymmetrical seams and/or rabbets in the insulation layer.

Option	Description
	Symmetrical seams. No rabbet for the insulation layer.
	Symmetrical seams. Rabbet depth can be defined for the insulation layer.
	Asymmetrical seams for the insulation layer and outer shell. No rabbet for the insulation layer.
	Asymmetrical seams for the insulation layer and outer shell. Rabbet depth can be defined for the insulation layer.

Offset / Mirror



You can define offset for the seam.

Rabbets tab

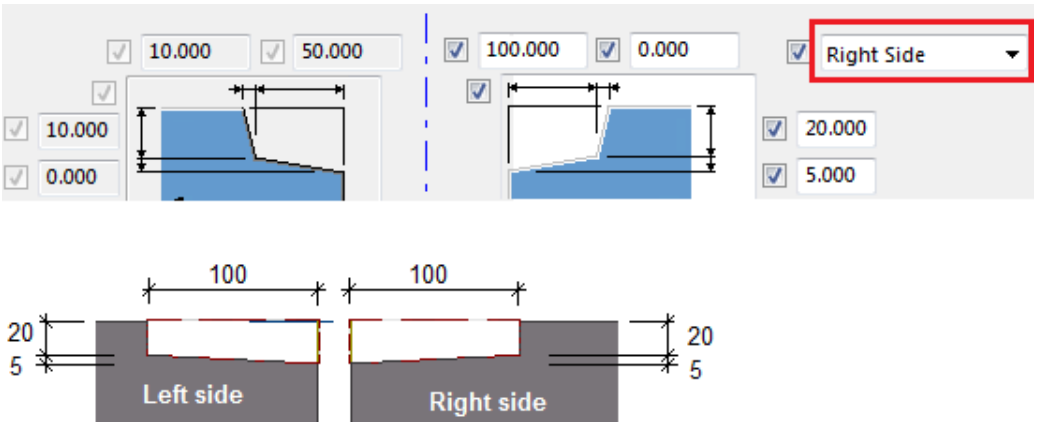
Use the **Rabbets** tab to define rabbets in inner shells, insulation layers and outer shells.

Rabbet side

Select to which side of the sandwich wall the rabbets are applied.

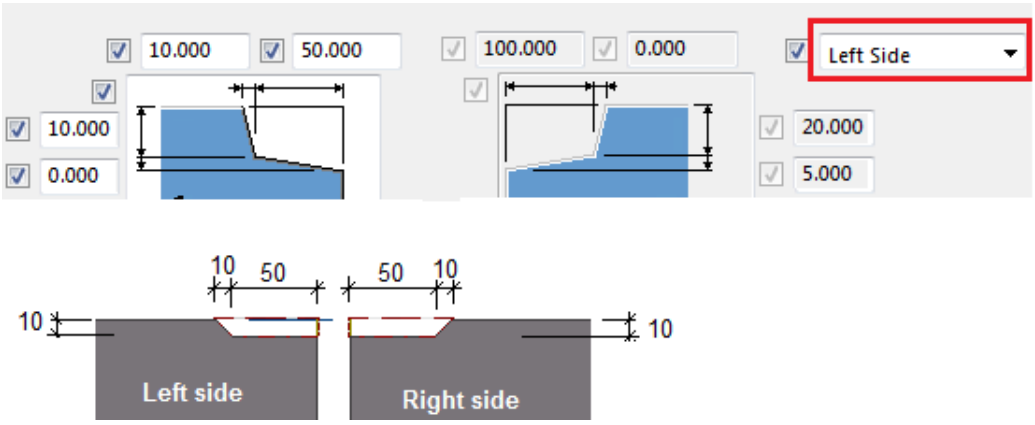
- **Right side**

  Rabbets are equal on both sides, but the rabbet dimensions on the right side are applied to both sides.



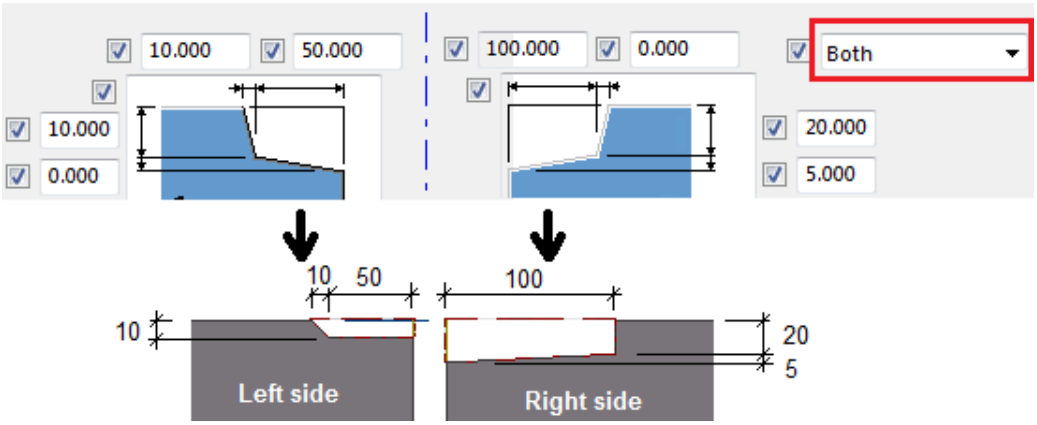
- **Left side**

  Rabbets are equal on both sides, but the rabbet dimensions on the left side are applied to both sides.



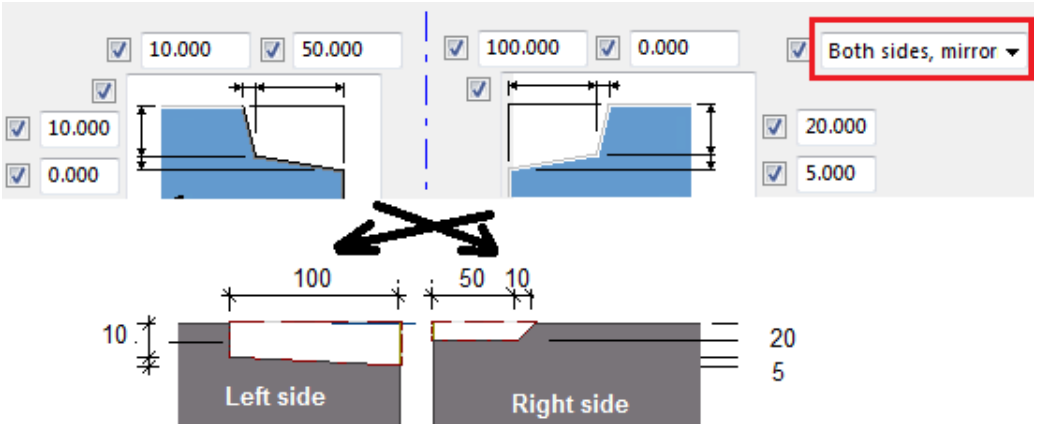
- **Both sides**

  Rabbet dimensions can be defined separately for both sides.



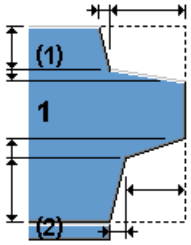
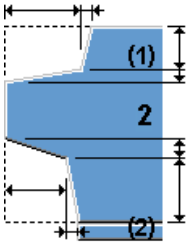
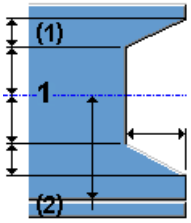
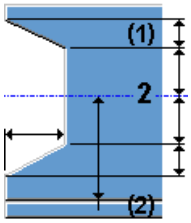
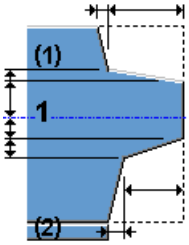
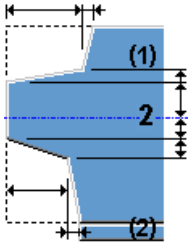
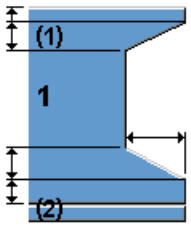
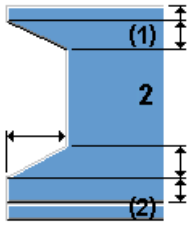
- **Both sides mirrored**

  Rabbet dimensions can be defined separately for both sides, but sides are mirrored.



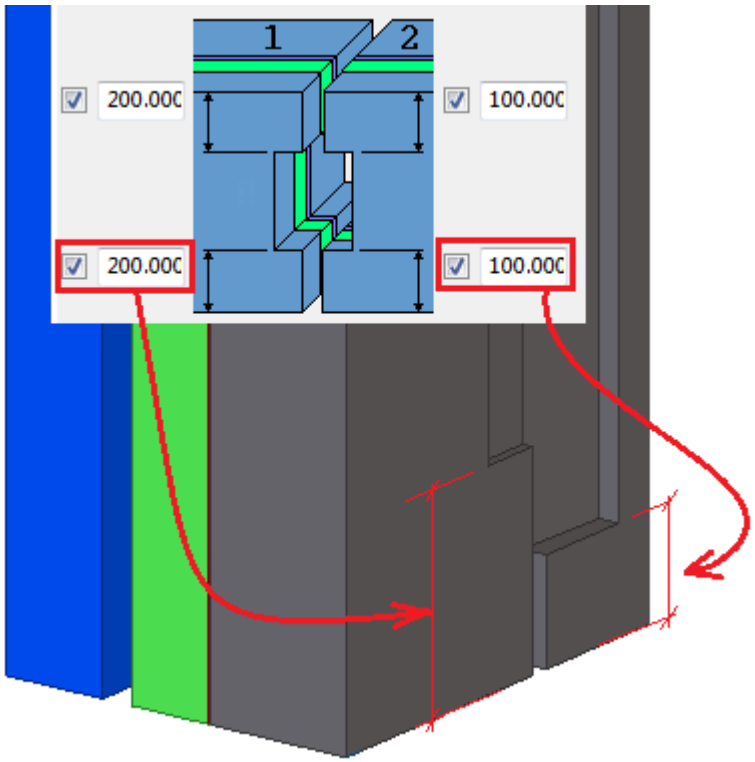
Connection type

You can select a tongue or groove joint connection for both sides.

Option	Option	Description
		Tongue joint connection, based on the outer geometry
		Groove joint connection, based on the center line
		Tongue joint connection, based on the center line
		Groove joint connection, based on the outer geometry

Rabbet position

Typically the vertical seams go from the top to the bottom of the sandwich wall. Define the seam offsets for the top and bottom sides. Offsets are applied only to the inner and outer shells.



General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Sandwich wall window

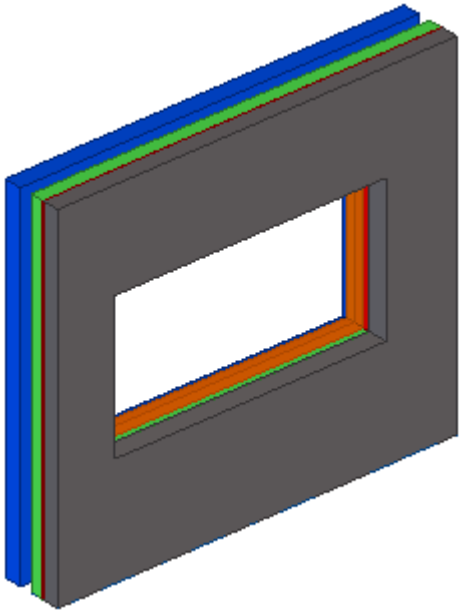
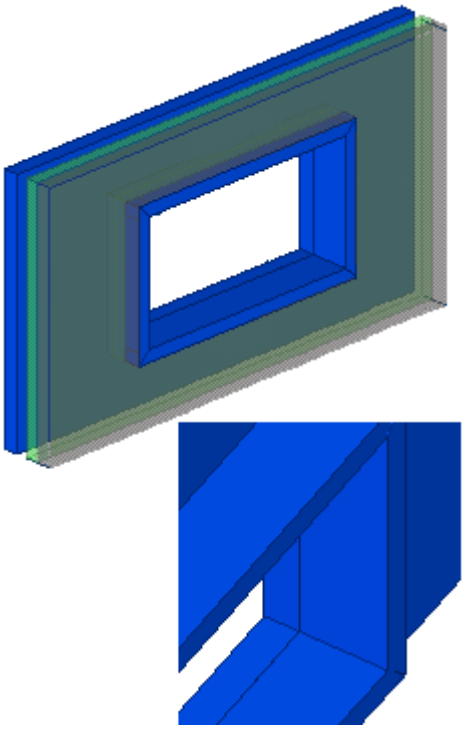
Sandwich wall window creates a rectangular window opening in a sandwich wall, or alternatively, a window and a door opening. The opening is created through up to four parts (inner shell, foil, insulation, and outer shell). You can select whether the foil is created. A frame can be created, as well as extra foils and additional parts. The frame can be a wooden frame or a concrete border, added to the inner shell.

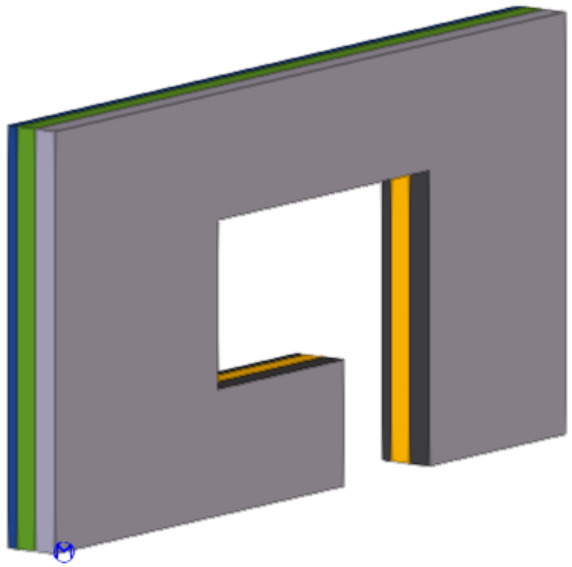
Objects created

- Rectangular opening
- Wooden frame or concrete border
- Extra foils

- Additional parts

Use for

Situation	Description
	Opening in a sandwich wall, with a wooden frame and foil layers.
	Opening in a sandwich wall, with a concrete border in the inner shell.

Situation	Description
	Opening in a sandwich wall, with a door opening.

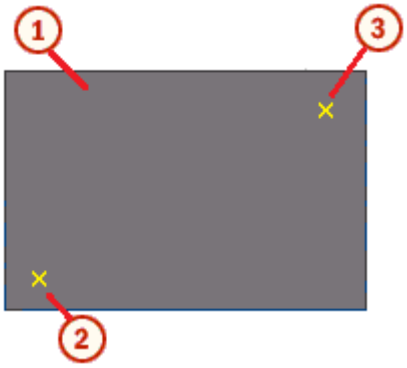
Selection order

1. Select the inner shell of the sandwich wall.
2. Pick the first position.
3. Pick the second position.

   The second position point is needed to complete the input also if you have selected an option for only one input point in the component dialog box. In this case, the second point location does not matter.

   The opening is created automatically when the second position is picked.

Part identification key

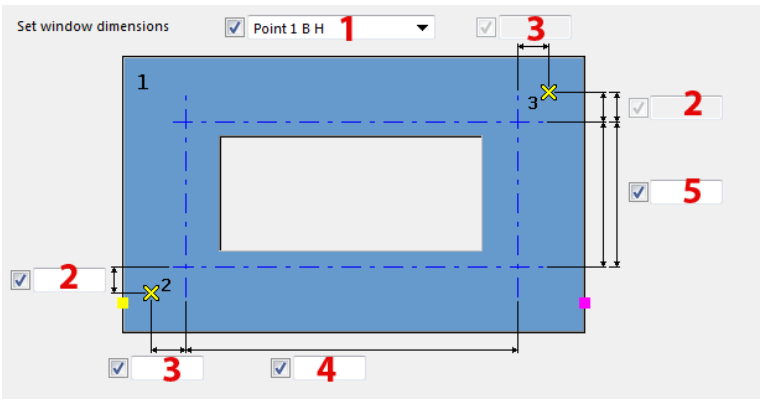
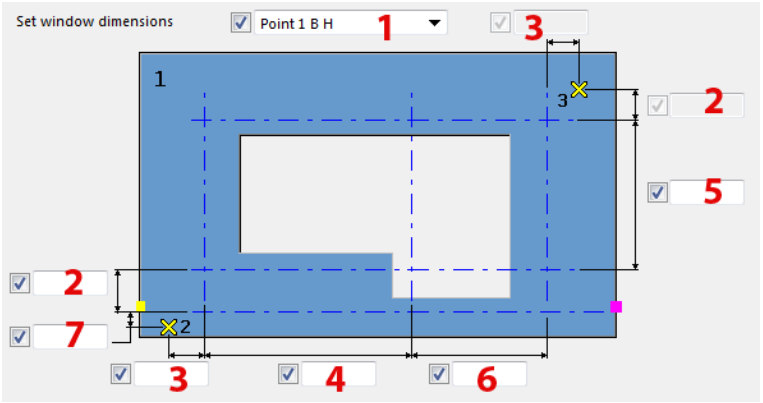


	Part
1	Sandwich wall element
2	First picked position

	Part
3	Second picked position

Picture tab
Use the **Picture** tab to control the dimensions and parts of the opening.

Dimensions

Option	Dimensions
Create **Window**	
Create **Window + Door** Set the **Door location** to **Right side** or **Left side**. The options on the **Door side detail** tab and **Door bottom detail** tab are available when you set the **Create** option to **Window + Door**.	

Option	Description
1	Set the points for the opening: • **2 points** Pick two points to create the opening (points 2 and 3 in the image). • **Point 1 B H** Pick two points to create the opening. With this option, point 2 shown in the image is the reference point. Point 3 is only

Option	Description
	needed to complete the input. Define the width and height of the opening. • **Point 2 B H** Pick two points to create the opening. With this option, point 3 shown in the image is the reference point. Define the width and height of the opening. • **2 Points H** Pick two points to create the opening (points 2 and 3 in the image). Define the height of the opening.
2	Vertical offset from the insertion point. When you create **Window + Door**, this is the vertical offset for the window.
3	Horizontal offset from the insertion point.
4	Width of the opening. When you create **Window + Door**, this is the width of the window.
5	Height of the window.
6	Width of the door.
7	Vertical offset for the door.

Select other layers

Option	Description
Insulation, **Wall**	Parts where the opening is created. To create the opening to the insulation and the external layer parts, enter the class numbers. If you do not enter any numbers, the opening is only created to the part you selected when applying the component.
Parts not in cast unit	Select **Yes** to create the opening to the parts that do not belong to the cast unit.
Foil	You can use **Sandwich wall window** both for sandwich panels with a foil and for sandwich panels without a foil (default). If you have a sandwich panel with a foil, select **Yes** and enter the class number of the foil. The frame shape options shown on the **Bottom detail**, **Left detail**, **Right detail**, and **Top detail** tabs depend on whether the foil is created or not.

Bottom detail tab / Left detail tab / Right detail tab
Use the **Bottom detail** tab, **Left detail** tab, and **Right detail** tab to control the size, position and shape of the opening bottom, left side, and right side, and the frame size and properties. On the **Right detail** tab, you can select that the right detail is created the same as the left detail.

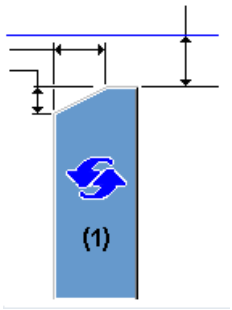
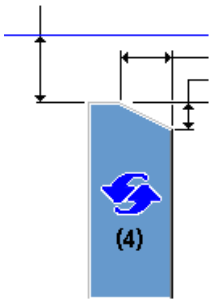
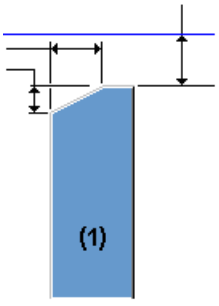
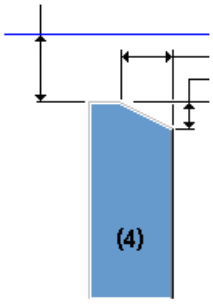
Frame

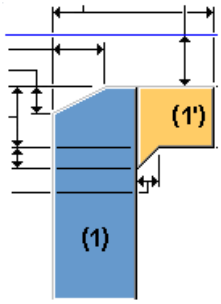
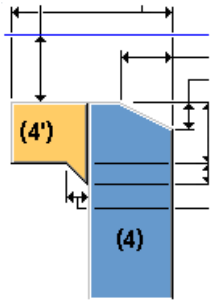
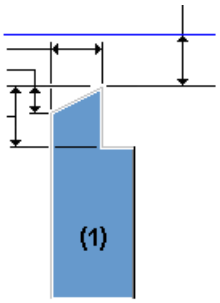
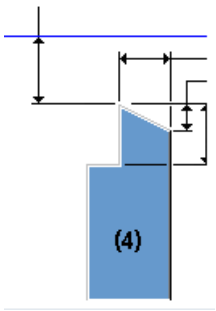
Part	Description	Default
Frame	Define the frame profile by selecting the profile from the profile catalog.	`50*50`

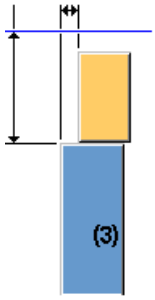
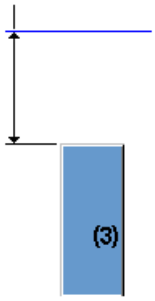
Option	Description
Pos_No	Prefix and start number for the part position number. You can enter the assembly position number on the second row.
Material	Material grade.
Name	Name that is shown in drawings and reports.
Class	Part class number.

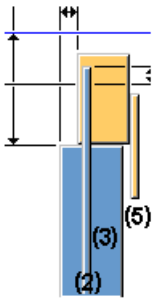
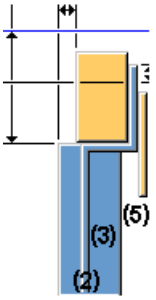
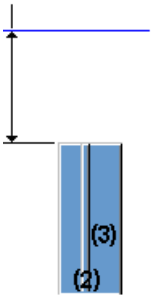
Frame shape

The options for inner and outer frame are the same regardless of whether the foil is created or not.

Inner	Outer	Description
		Default Cut or extension is not created. AutoDefaults can change this option. Any of the three options is used depending on the frame size.
		Cut or extension is not created.

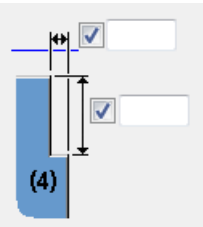
Inner	Outer	Description
		Extension is created. If the frame is narrower than the insulation, the concrete part is extended to fill the gap.
		Cut is created. If the frame is wider than the insulation, the concrete part is cut so that the frame fits.

Middle	Description
	This option is available when the **Foil** option is set to **Do not create** on the **Picture** tab. Frame is created.
	This option is available when the **Foil** option is set to **Do not create** on the **Picture** tab. Frame is not created.

Middle	Description
	This option is available when you select **Yes** in the **Foil** option on the **Picture** tab. Frame is created.
	This option is available when you select **Yes** in the **Foil** option on the **Picture** tab. Frame is created.
	This option is available when you select **Yes** in the **Foil** option on the **Picture** tab. Frame is not created.

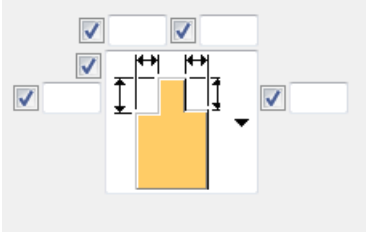
Additional recess

Define the vertical and horizontal dimensions for the additional recess. By default, the recess is not created.



Frame corner cuts

Option	Description
	Frame corners are not cut.

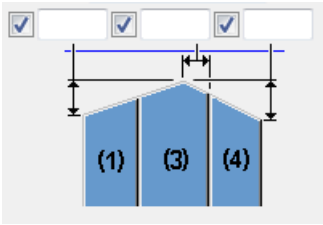
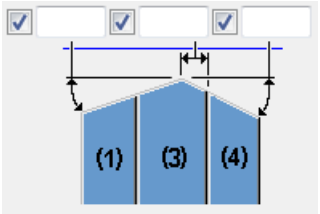
Option	Description
	Define the horizontal and vertical dimensions of the frame corner cuts.

Create filler parts as insulation

Select whether filler parts are created as shells or as insulation.

Continuous slope

Select whether frame edges are sloped.

Option	Description
	**By length**
	**By angle**

You can set the length or the angle for both sides separately.

The default is that frame edges are not sloped.

Top detail tab

Use the **Top detail** tab to control the size, position, number, and shape of the opening top side.

Part

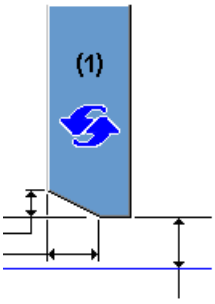
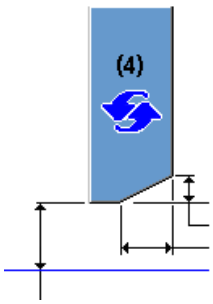
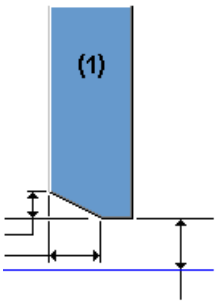
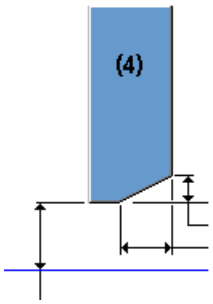
Part	Description	Default
Frame	Define the frame profile by selecting the profile from the profile catalog.	50*50

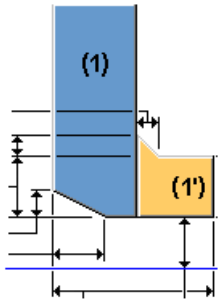
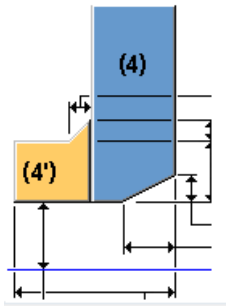
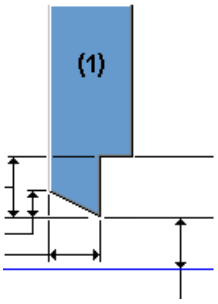
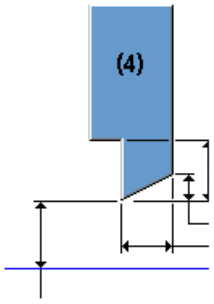
Part	Description	Default
Foil2	**Foil2** is the additional foil created only on the top side. Define the thickness.	

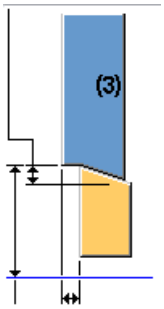
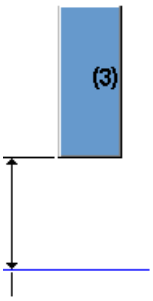
Option	Description
Pos_No	Prefix and start number for the part position number. You can enter the assembly position number on the second row.
Material	Material grade.
Name	Name that is shown in drawings and reports.
Class	Part class number.

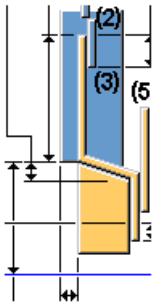
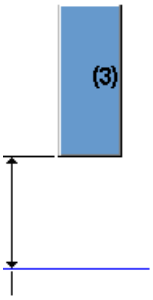
Top frame shape

The options for inner and outer frame are the same regardless of whether the foil is created or not.

Inner	Outer	Description
		Default Cut or extension is not created. AutoDefaults can change this option. Any of the three options is used depending on the frame size.
		Cut or extension is not created.

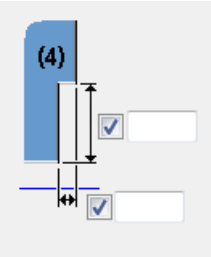
Inner	Outer	Description
		Extension is created. If the frame is narrower than the insulation, the concrete part is extended to fill the gap.
		Cut is created. If the frame is wider than the insulation, the concrete part is cut so that the frame fits.

Middle	Description
	This option is available when the **Foil** option is set to **Do not create** on the **Picture** tab. Frame is created.
	This option is available when the **Foil** option is set to **Do not create** on the **Picture** tab. Frame is not created.

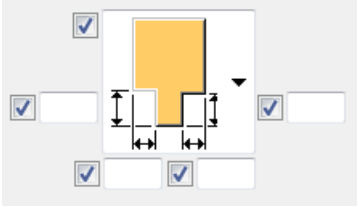
Middle	Description
	This option is available when you select **Yes** in the **Foil** option on the **Picture** tab. Frame is created.
	This option is available when you select **Yes** in the **Foil** option on the **Picture** tab. Frame is not created.

Additional recess

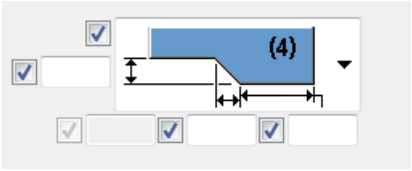
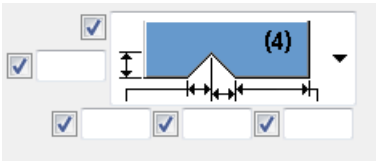
Define the vertical and horizontal dimensions for the additional recess. By default, the recess is not created.



Frame corner cuts

Option	Description
	Frame corners are not cut.
	Horizontal and vertical dimensions of the frame corner cuts.

Drip mold

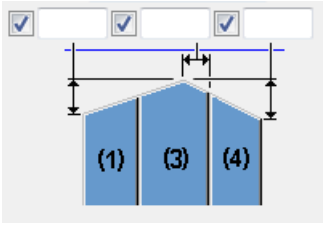
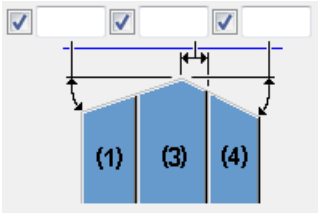
Option	Description
	Drip mold is not created.
	Drip mold is created. Define the mold dimensions.
	Drip mold is created. Define the mold dimensions. The drip mold can be created to any of the shape options. For example, if the shape has a bevel, the drip mold is created at the same angle as the bevel.

Create filler parts as insulation

Select whether filler parts are created as shells or as insulation.

Continuous slope

Select whether frame edges are sloped.

Option	Description
	**By length**
	**By angle**

You can set the length or the angle for both sides separately.

The default is that frame edges are not sloped.

Extra foils tab

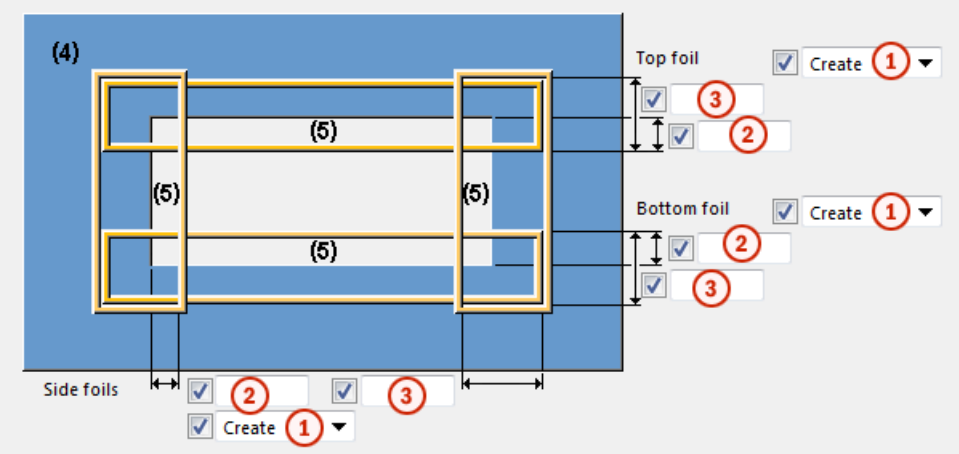
Use the **Extra foils** tab to control the creation of extra foils on the top, bottom, right side, and left side of the opening.

Part

Part	Description	Default
Extra foil	Thickness of the foil.	50*50

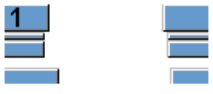
Option	Description
Pos_No	Prefix and start number for the part position number. You can enter the assembly position number on the second row.
Material	Material grade.
Name	Name that is shown in drawings and reports.
Class	Part class number.

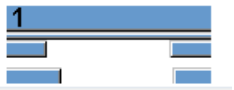
Extra foil dimensions



	Description
1	Define whether extra foils are created.
2	Define the inside offset.
3	Define the foil width.

Shells to cut

Option	Description
	All shells are cut.

Option	Description
	The inner shell is not cut.
	The inner shell and the foil are not cut.
	The inner shell and the foil are not cut. An insulation extension is created. Define the width and the material grade of the insulation extension.
	The outer shell is cut.

Additional parts tab

Use the **Additional parts** tab to control the size, position, number, and alignment of additional parts.

Parts

Select whether additional parts, custom component parts, or no additional parts are created.

Part	Description	Default
Parts	Define the profile of the additional parts by selecting the profile from the profile catalog.	D5
Custom component parts	Define the custom component parts by selecting the component from the **Applications & components** catalog. You can also use a configuration file to define the properties.	

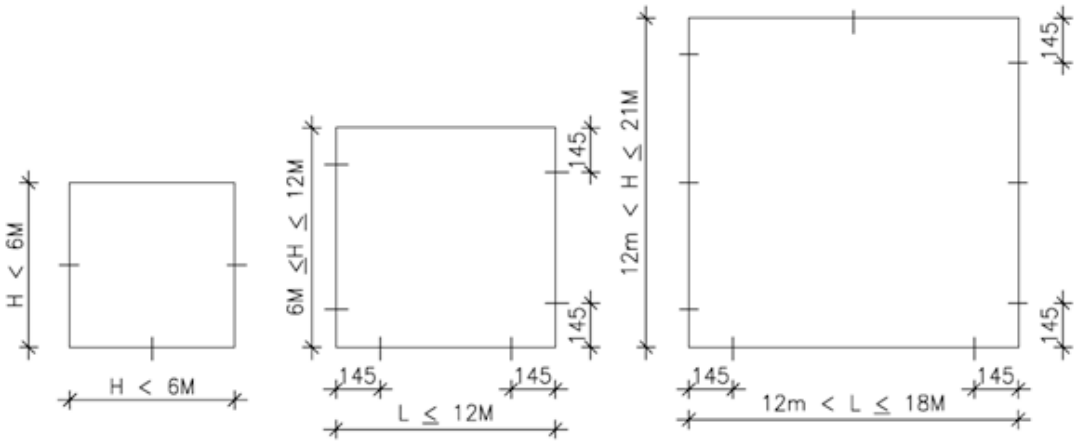
Option	Description
Pos_No	Prefix and start number for the part position number. You can enter the assembly position number on the second row.
Material	Material grade.
Name	Name that is shown in drawings and reports.
Class	Part class number.

Define distances

Select whether to define the distances between the parts by entering the distance dimensions on this tab or by using an external text file. You can use an external file to automate the creation of additional parts when window sizes differ.

The name of the external file is `SandwichPanelWindowAdditionalParts.dat`. The file is located in the `..\Environments\Common\system folder`.

The image below shows an example of a local standard for creating additional parts. M is a module that is 100 mm.



The `SandwichPanelWindowAdditionalParts.dat` configuration file that contains the needed specifications for the example above is as follows:

```
BOTTOM;
0;599;1;1
600;1200;1;2;145
1201;1800;1;3;145
TOP;
1201;1800;1;1
LEFT;
0;599;1;1
600;1200;1;2;145
1201;2100;1;3;145
RIGHT;
0;599;1;1
600;1200;1;2;145
1201;2100;1;3;145
```

The keywords `BOTTOM`, `TOP`, `LEFT` and `RIGHT` define the side of the window the settings are applied to.

The syntax in the file is as follows: `bmin;bmax;cm;nd;dmax;d1;d2;d3`.

`bmin`	Smallest window dimension to apply the setting.
`bmax`	Biggest window dimension to apply the setting.
`cm`	Creation method: • `1` means use the number of parts for `nd`.

	• 2 means use maximum spacing for nd.
nd	Depending on the creation method (cm): • Number of parts, when cm is 1. • Maximum allowed distance between parts, when cm is 2.
d1	Distance from the start point to the first additional part (optional).
d2	Distance from the first additional part to the second (optional).
d3	Distance from the second additional part to the third (optional).

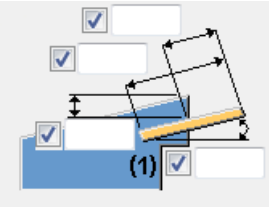
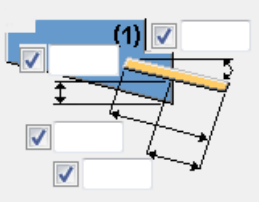
The additional parts are placed symmetrically, so distances d1, d2 and d3 are also used from the end point. If the optional distances d1, d2 and d3 are not defined, the parts are placed at equal distances.

In the image above 0;599;1;1 means that when the window width is equal or bigger than 0, but equal or less than 599, the creation method is by number of parts and 1 part should be created.

600;1200;2;300;145 would mean that when the window width is equal or bigger than 600, but equal or less than 1200, the creation method is use maximum spacing. The maximum allowed spacing between additional parts is 300 mm. The first part (and last part) distance is 145 mm. If the window width is 1200, three more parts are needed (1200 - 2x145)/300=3.03. The spacing between the additional parts will then be (1200 - 2x145)/4 = 227.5 mm.

Placement

Select whether additional part placement is the same or different on each side.

Option	Description
	Placement dimensions for the bottom, left side, and right side. If additional part placement is the same on each side, define the bottom dimensions only.
	Placement dimensions for the top side.

Door edge options

The options for controlling door edges are available when you set the **Create** option to **Window + Door** on the **Picture** tab, and the **Placement** option to **Different each side** on the **Additional parts** tab.

Option	Description
Create on door side	**No**, **Same as window side**, **Same as door side**
Create in bottom detail	**No**, **Same as bottom**

Connections tab

Use the **Connections** tab to control how the wooden frame, additional parts, extra foil, and extensions are connected to the shells.

Add

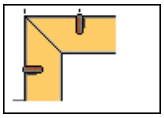
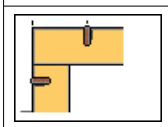
Option	Description
Wooden frame to, **Additional parts**, **Extra foil to**	Select to which part the wooden frame, additional parts, and extra foil are connected. You can create the frame as an assembly. To add window and door frames to a shell as one subassembly, select **Subassembly** for **Wooden frame to**. The **Extra foil to** option is shown if you have selected to create the foil on the **Picture** tab.

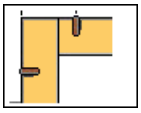
Connect extensions to their shells

Option	Description
Inside shell extension, **Outside shell extension**, **Insulation shell extension**, **Foil extension**	Select how the extension parts are connected to their main parts. The **Foil extension** option is shown if you have selected to create the foil on the **Picture** tab.

Edge connection

Define the corner connection type for wooden frames.

Option	Description
	Cut both frames in a 45-degree angle.
	Extend horizontal frames.

Option	Description
	Extend vertical frames.

Door side detail tab / Door bottom detail tab
Use the **Door side detail** tab and **Door bottom detail** tab to control the size, position, and shape of the door side and bottom, and the frame size and properties.

Create door

The options on the **Door side detail** tab and **Door bottom detail** tab are available when you set the **Create** option to **Window + Door** on the **Picture** tab.

Same as options

- **Same as** on the **Door side detail** tab:

  Select the door short side detail.

  - **None**
  - **Same as window side** follows the side setting of the window side.
  - **Same as door side** follows the door side setting.

  The door long side detail follows the setting of the corresponding window side.
- **Same as bottom** on the **Door bottom detail** tab:

  Yes follows the setting of the bottom window.

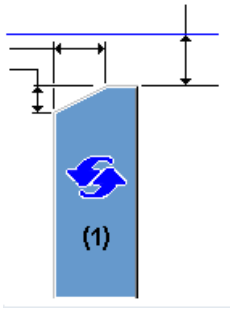
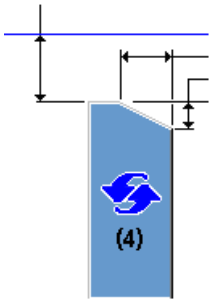
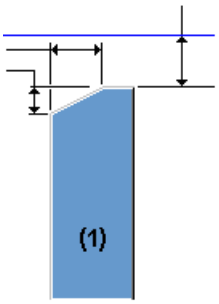
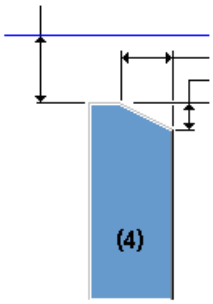
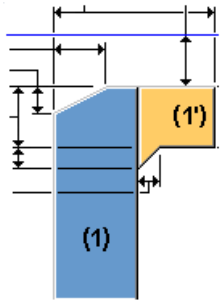
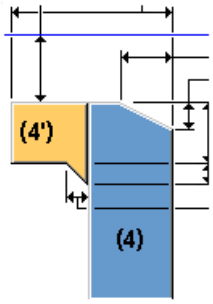
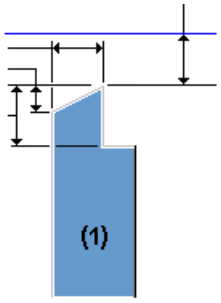
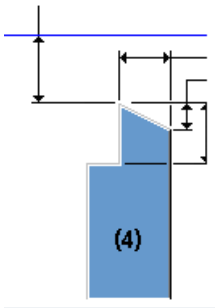
Frame

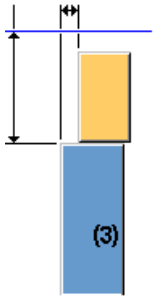
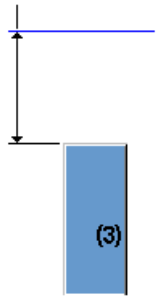
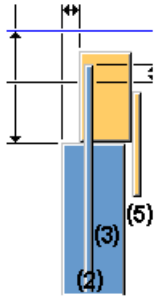
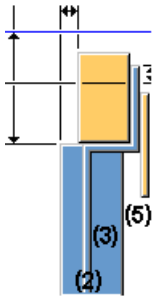
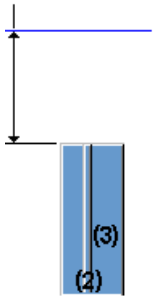
Part	Description	Default
Frame	Define the frame profile by selecting the profile from the profile catalog.	`50*50`

Option	Description
Pos_No	Prefix and start number for the part position number. You can enter the assembly position number on the second row.
Material	Material grade.
Name	Name that is shown in drawings and reports.
Class	Part class number.

Frame shape

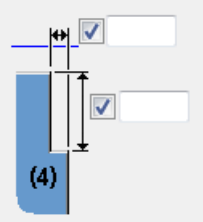
The options for inner and outer frame are the same regardless of whether the foil is created or not.

Inner	Outer	Description
		Default Cut or extension is not created. AutoDefaults can change this option. Any of the three options is used depending on the frame size.
		Cut or extension is not created.
		Extension is created. If the frame is narrower than the insulation, the concrete part is extended to fill the gap.
		Cut is created. If the frame is wider than the insulation, the concrete part is cut so that the frame fits.

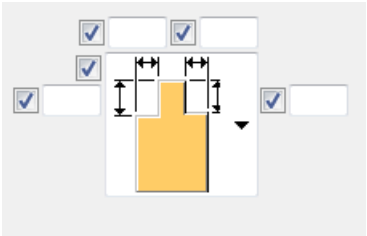
Middle	Description
	This option is available when the **Foil** option is set to **Do not create** on the **Picture** tab. Frame is created.
	This option is available when the **Foil** option is set to **Do not create** on the **Picture** tab. Frame is not created.
	This option is available when you select **Yes** in the **Foil** option on the **Picture** tab. Frame is created.
	This option is available when you select **Yes** in the **Foil** option on the **Picture** tab. Frame is created.
	This option is available when you select **Yes** in the **Foil** option on the **Picture** tab. Frame is not created.

Additional recess

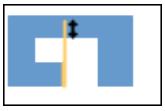
Define the vertical and horizontal dimensions for the additional recess. By default, the recess is not created.



Frame corner cuts

Option	Description
	Frame corners are not cut.
	Define the horizontal and vertical dimensions of the frame corner cuts.

Door side frame extension

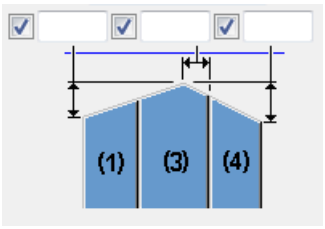
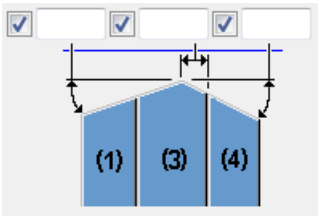
Option	Description
	Frame on the door side is created only to the bottom level of the window.
	Frame on the door side is extended to the top level of the window. Enter the extension distance in the box.

Create filler parts as insulation

Select whether filler parts are created as shells or as insulation.

Continuous slope

Select whether frame edges are sloped.

Option	Description
	**By length**
	**By angle**

You can set the length or the angle for both sides separately.

The default is that frame edges are not sloped.

Wall layout tools

Wall Layout tools is a set of components that you can use for creating and modifying all common types of concrete walls, such as solid precast panels from single layers to double walls and sandwich walls, and different wall structures that are cast on the site. The wall structure may contain several layers, for example, structural layers, insulation, void, and surface treatments. You can use direct modification to flexibly change the wall geometry, layer offsets, openings, and seam lines.

Wall layout is the main component in the set and it is used for defining the wall layout. **Wall layout** is available on the **Concrete** tab, click **Panel** --> **Wall layout**, and in the **Applications & components** catalog.

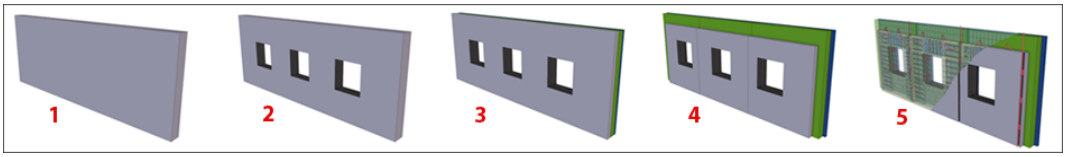
The other **Wall Layout** tools are available in the **Applications & components** catalog:

- **Wall layout connector** connects the walls to each other.
- **Wall layout T connector** connects walls to each other with T connections. **Wall layout T connector** can also create a seam at the connection location.
- **Wall layout opening** sets the default values for openings. Use direct modification to create and modify openings.
- **Wall layout seam** divides the wall segment into two or more cast units by adding seams. Use direct modification to create seams.
- **Wall layout elementation** divides the wall into elements according to length, height, number, weight, or crane lifting capacity.
- **Wall layout layer swapper** is intended for swapping the casting order of layers in a double wall. Only certain properties of the two layers are

swapped, such as name, class, numbering settings, and all UDAs. The geometry of the layers is not swapped. If one layer is created using the layer creation option **Add as sub-assembly** (shell 2), and the other layer is defined as **Add to cast unit** (shell 1), these properties are also swapped.

Wall layout

Wall layout creates a single wall or multiple straight wall segments. The wall can be a single-layer solid wall, a double wall, or a sandwich wall that has any number of layers.

Design workflow with Wall layout tools

1. You can build a shape. 2. You can build a shape and create openings. 3. You can create wall type definitions. 4. You can create wall elementation and define detailed wall geometry. 5. You can define detailed connections, openings, embeds, and reinforcement.

Selection order

1. Ensure that the **Direct modification** switch is active.
2. On the **Concrete** tab, click **Panel** --> **Wall layout**.
3. Select the appropriate command on the contextual toolbar:

   - To create one or more straight wall segments, click , and then pick two or more points.

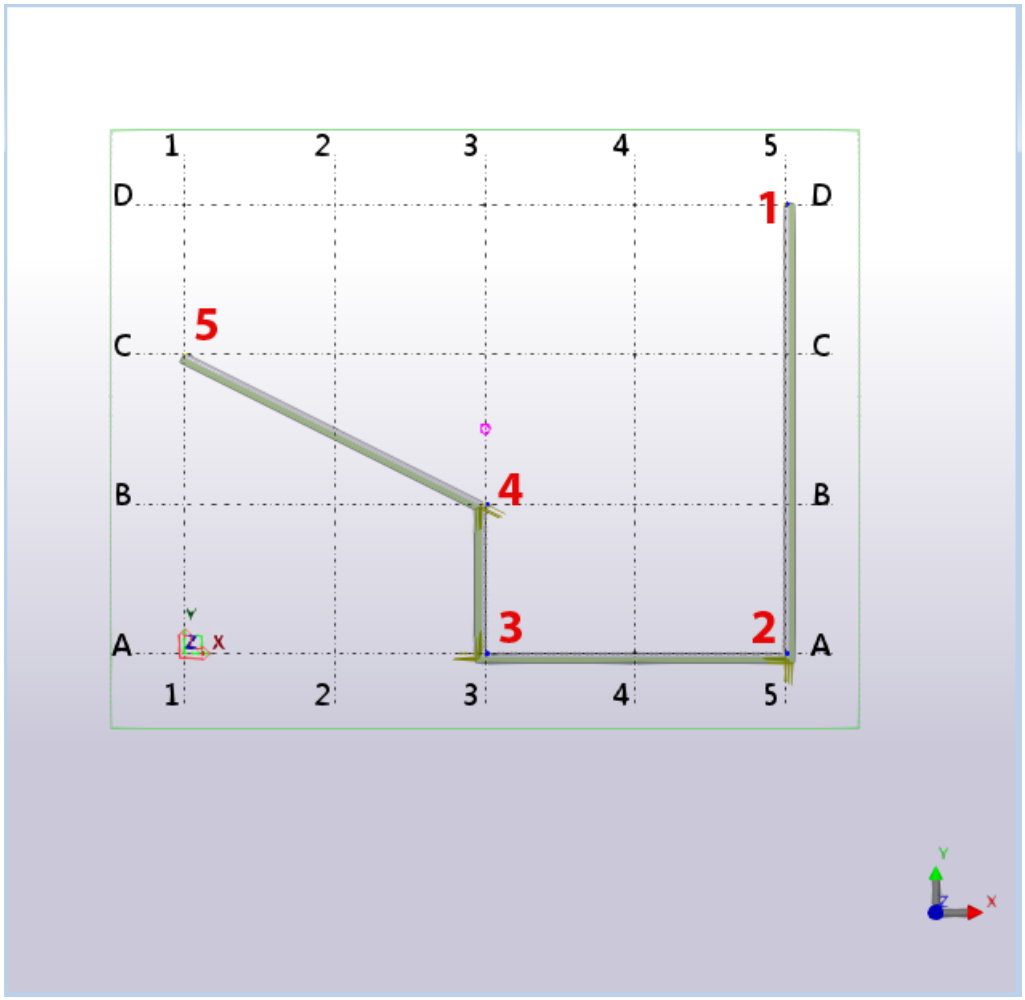
     Note that you can finish the creation to the first picked point to create a closed wall structure.

   - To create the wall layout as a closed wall, click , and then pick two or more points.

     Wall layout will automatically connect the first and the last point that you have picked.
4. Click the middle mouse button to create the wall.

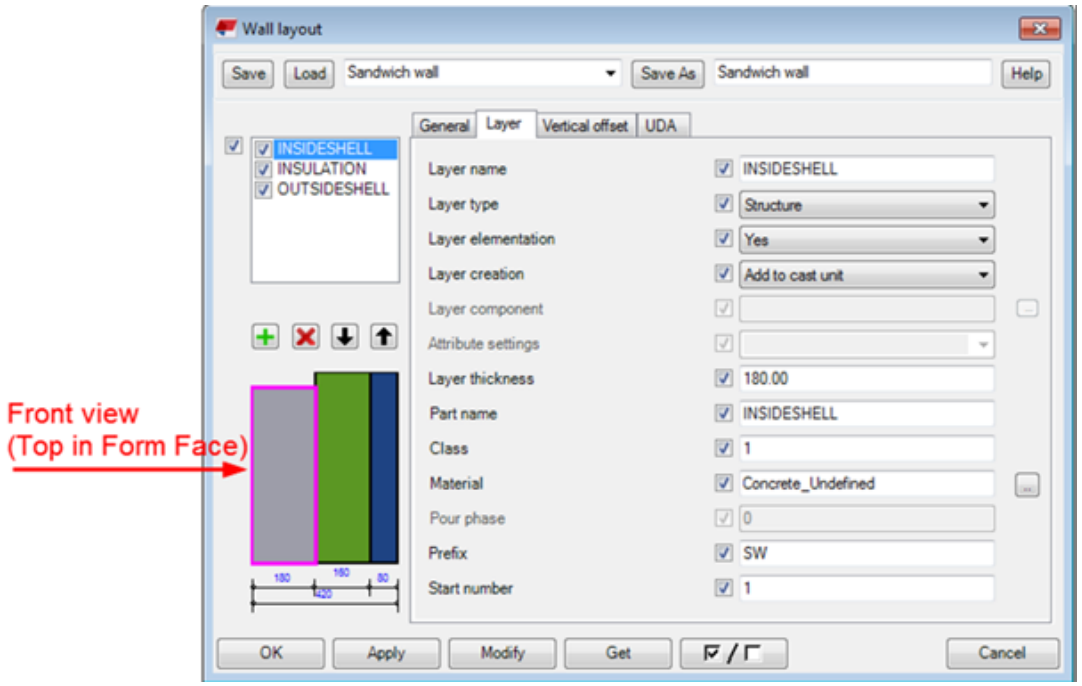
Depending on how many points you pick and how you pick the points, one or more straight wall segments or a closed wall is created in the model. Each created straight segment is a separate instance of the **Wall layout** component that you can modify separately. When you create two or more wall segments, an instance of the **Wall layout connector** component is added between each wall segment.

In the image below, there are four wall segments in an open shape between the picked points, 1 - 5, and three wall layout connectors at points 2, 3 and 4. If you had created a closed wall, there would be a fifth wall segment between points 5 and 1.



The modeling direction of the wall layout determines which face of the wall layout is shown in the front view of a cast unit drawing when the cast unit drawing coordinate system is set to **Fixed**.

In the image below, the top-in-form face is not set by **Wall layout**. The image shows an example of sandwich wall settings that you should use when the outside shell is cast against the pallet, and the inside shell is the uppermost shell in the casting.



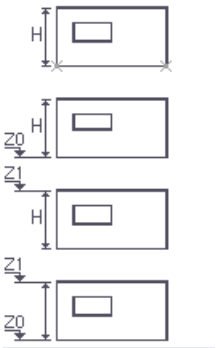
Alternatively, you can set the top-in-form face with **Wall layout**. Set `Walllayout.Udas.dat` to have the top-in-form face property by adding the following row:

```
option FixedMainView j_FixedDrawingMainView
```

The `Walllayout.Udas.dat` file can be located in the model folder or any of the system folders.

General tab

Use the **General** tab to define the basic properties for the whole **Wall layout** component. The properties on all other tabs are layer-specific, and apply to the layer that you have currently selected in the layer list.

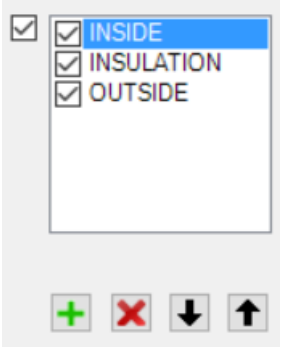
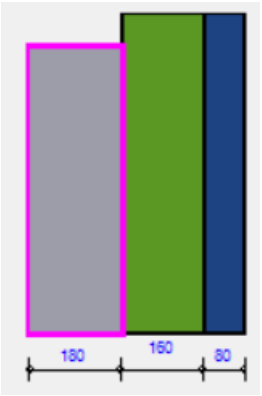
Option	Description
H, **Z1**, **Z0** 	You have four options for defining the height and the global Z coordinates: • Define the wall bottom level according to the creation points, and the **Position in depth** option and value. Define the height with the **H** option. • Define the wall bottom level with the **Z0** option (global Z coordinate), and the height with the **H** option. • Define the wall top level with the **Z1** option (global Z coordinate), and the height with the **H** option.

Option	Description
	• Define the wall bottom level with the **Z0** option (global Z coordinate), and the wall top level with the **Z1** option (global Z coordinate). You can use the **H** option to modify the height of rectangular walls. If you change the wall shape so that it is not rectangular, you cannot define height with this option. Use direct modification instead.
Total thickness	Enter the total thickness of the wall layout. Layer thickness is calculated based on the total thickness. Use the total thickness for double walls where the thickness of the void layer or the CIP layer is often flexible. This means that **Wall layout** can adjust the thickness so that the total thickness is as defined. • If there are no flexible layers in the wall layout, you can leave the total thickness option empty. Total thickness is not used if there are no flexible layers. • If there is more than one flexible layer, the thickness is divided equally between those layers.
Cast unit name	Enter a name for the cast unit.
Prefix	Enter a prefix for the cast unit.
Start number	Enter a start number for the cast unit.
Main part layer	Select which layer part is set as the main part of the cast unit. The **Default** option sets the structural part with the greatest volume as the main part.
Position in plane	Define the wall layout location in relation to the line between the input points. Enter the offset distance, if needed.
Position in depth	Define the wall layout vertical location in relation to the line between the input points. Enter the offset distance, if needed.

Layer tab

Use the **Layer** tab to define the properties of a single layer. Select the layer in the layer list, or in the preview image.

Layer list

Layers	Description	Example
	The layer list shows the layers of the wall. Use the buttons to control the number and the order of the layers: • Click to add a new layer. • Click to remove the selected layer. • Click to change the order of the layers by moving the selected layer up or down in the list. Use the check box in front of the layer name to control whether the properties of the selected layer are modified. Use the leftmost check box to control whether the number of layers and the order of the layers are modified when you modify the **Wall layout** component.	The example image below shows a preview of the wall layers. The image is automatically updated when you change the layer type. The selected layer is highlighted with a magenta frame. 

Layer properties

Option	Description
Layer name	Enter a name for the layer. This name is shown in the layer list. Layer names are also shown on the contextual toolbar when you modify wall boundary offsets. You can select which layer to modify. Layer names are not visible in reports or drawings.
Layer type	• **Structure**: Typically used for solid walls, double walls, and the concrete layers of sandwich walls. Use

Option	Description
	this type if you want to create a cast-in-place layer between double wall layers instead of a void. • **Insulation**: Insulation layer in a sandwich wall. Insulation gets a different density in **Wall layout elementation** (100 kg/m³). • **Void**: Void between double wall layers. Use this type if you do not want to create any parts for the layer. • **Foil**: Typically a very thin layer which is not detailed in the corners. • **Surface**: Surface treatment on top of a structural layer. Typically, a surface layer is the first and/or the last layer. • **Component**: Layer is not created as a part. Instead, an instance of the component selected in the **Layer component** option is added.
Layer elementation	Select whether the layer is split into two pieces using seams.
Layer creation	Select how the layer is created to the precast cast unit: • **Add to cast unit**: All layer parts are added to a single main cast unit. • **Add as sub-assembly**: The layer part is added as a sub-assembly to the main cast unit. • **Do not add to cast unit**: The layer part is not added to the main cast unit. It will be its own assembly or cast unit. This option changes the concrete type from precast to **cast in place**.
Layer component	Select a component when you have set the layer type to **Component**. Instead of creating the layer as a part, an instance of the component is created. You can control some of the component properties from **Wall layout** properties. Define the property names in the component accordingly: • `P_Height` - Layer height (maximum height of the layer part) • `P_Thickness` - Layer thickness • `P_Name` - Layer part name • `P_Class` - Layer part class • `P_Material` - Layer material

Option	Description
	• `P_Prefix` - Layer part prefix • `P_StartNo` - Layer part start number The creation points of the component are in the same locations as when creating the layer as a part. If the custom part or component property names are not the same as the default names, you can define the parameters in the `xxxxx.LayerComponentInfo.xml` file where `xxxx` is the component name. In this file you can also specify the mapping for other layer properties so that the values given in the attribute file will be overridden with the layer properties defined in the **Wall layout** dialog box. For an example of the file, see the **Customize the wall layout** section.
Attribute settings	Select an attribute file for the layer component. If you need specific component properties for the layer component, you can define the properties in the component dialog box and save them as an attribute file.
Layer thickness	Enter the layer thickness. You can leave the layer thickness empty to let **Wall layout** set the layer thickness so that the total thickness is correct as defined on the **General** tab. The void layer or the CIP layer in a double wall can often have a flexible thickness.
Part name, **Class**, **Material**, **Pour phase**, **Surface treatment**, **Prefix**, **Start number**	Define the name, class, material, part prefix and start number of the layer part. Define the pour phase for CIP layers. Select the surface treatment when you have set the layer type to **Surface**.

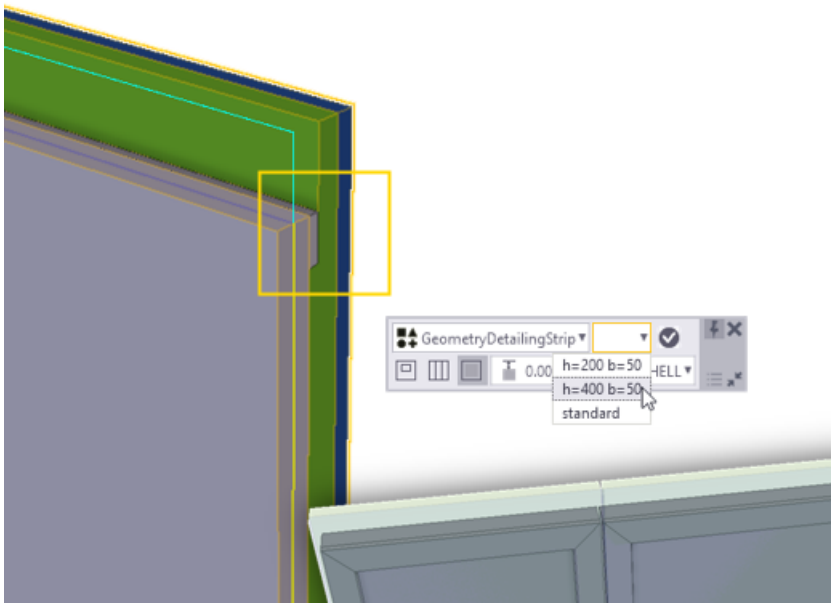
Vertical offset tab

Use the **Vertical offset** tab to define the vertical offsets of each layer at the top and bottom horizontal edges of the wall layout.

Note that you can define the offsets at any other edge by using direct modification and seams.

Option	Description
Offset	Define the vertical offset at each layer.
Component name	Select a component for detailing the offsets. For example, you can use the **Geometry detailing strip** component to create thickenings for sandwich walls. You can create different settings for **Geometry detailing strip** and select the appropriate settings for each situation.
Component attributes	Select the appropriate component settings.

You can also add detailing to an edge of a wall layout. Select the wall layout and activate the editing by selecting **Modify wall boundary offsets** on the contextual toolbar. Select the wall layout edge to which you want to add the component. Select the detailing component and the settings that you want to use, and click to confirm.



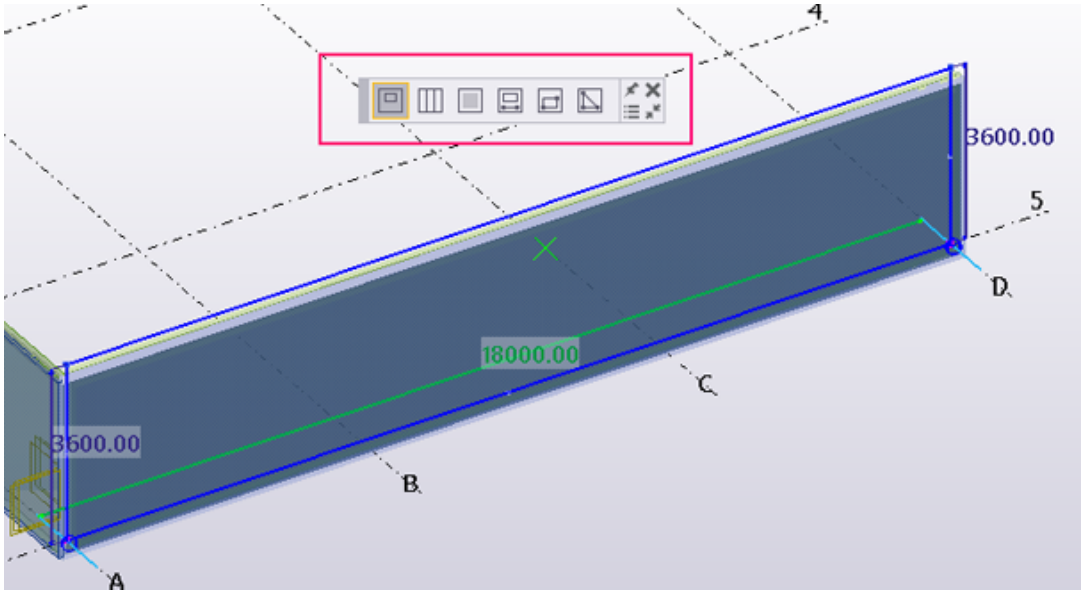
UDA tab

Use the **UDA** tab to define the user-defined attributes of each layer part in the wall layout.

Note that the content of the **UDA** tab may vary depending on your Tekla Structures environment. You can customize the content of the **UDA** tab.

Modify the geometry of the wall layout

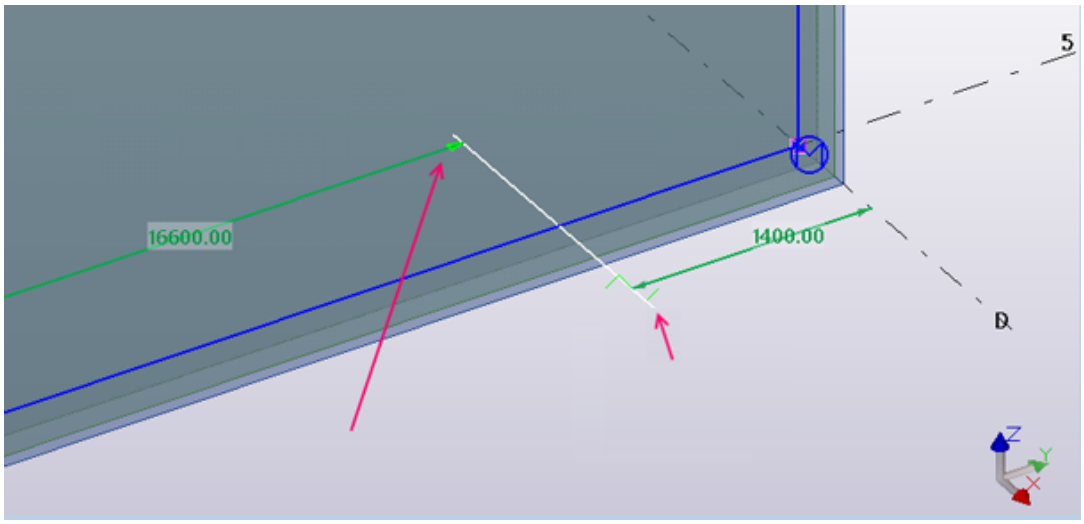
You can use direct modification to modify the wall layout. Before you start, ensure that the **Direct modification** switch is active. Select the wall layout to display the contextual toolbar. To modify the wall layout, select the appropriate command from the toolbar.



You can modify the outer edges of the wall layout by dragging the edges and corner handles. You can also insert vertices by dragging the edge midpoint handles, and delete vertices by selecting a corner handle or edge, and pressing the **Delete** key.

In addition to the standard polygon modification, you can drag the special line handles at the ends of the wall layout to make the wall layout shorter or longer. You can also drag the line handle perpendicular to the wall layout by holding down the **Alt** key while dragging.

As an alternative to dragging the line handle, you can drag the dimension arrowhead. If you know how much you want to shorten or lengthen the wall layout, select the dimension arrowhead and enter the value by which you want the dimension to change. Note that you can use the dimension arrowheads and line handles in a plan view whereas polygon modification is possible only in 3D or section views.



Modify the offset at layer edges

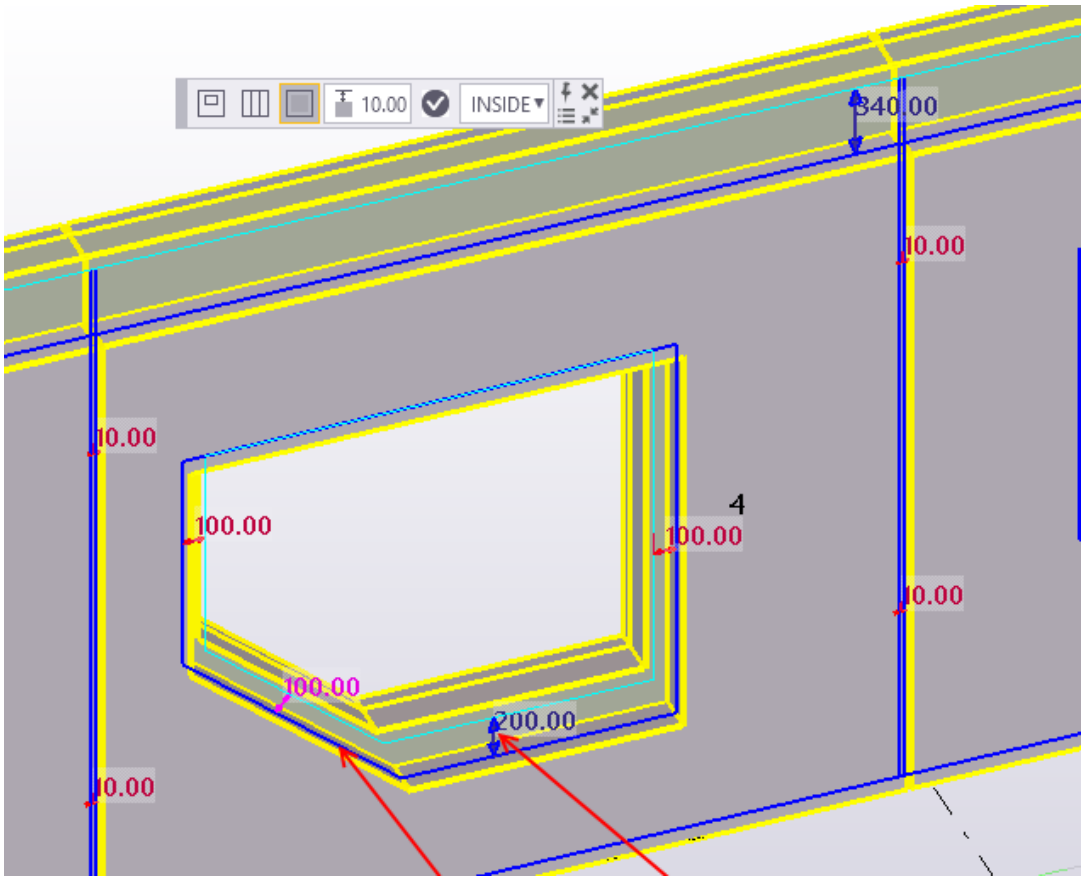
Use the **Modify wall boundary offsets** command on the contextual toolbar to modify the offsets of the layers at the outer edges of the wall layout, or at the edges of the openings or seams. Select the layer that you are going to modify from the list on the toolbar.

You have three options for modifying the offset at a particular edge:

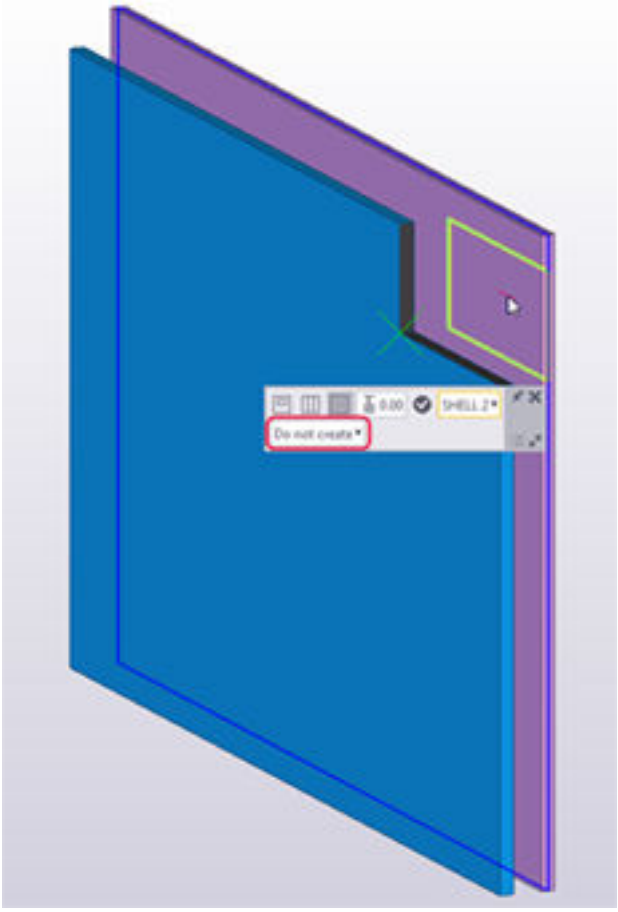
- Drag the line handle at the edge. You can only drag the line handle perpendicular to the edge.
- Drag the dimension arrowhead and enter the measurement value.
- Select the line handle and enter a new offset value.

  You can select two or more line handles by holding down the **Shift** key when selecting the handles.

You can modify the top and bottom offsets at the outer edges in the **Wall layout** dialog box, and the seam offsets in the **Wall layout seam** dialog box.



You can define that a certain layer is not cut with the opening. First select the opening. On the contextual toolbar, select the layer and then **Do not create**.

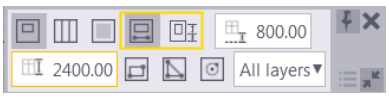


Wall layout opening
Use direct modification to add openings to wall layouts. Before you start, ensure that the **Direct modification** switch is active.

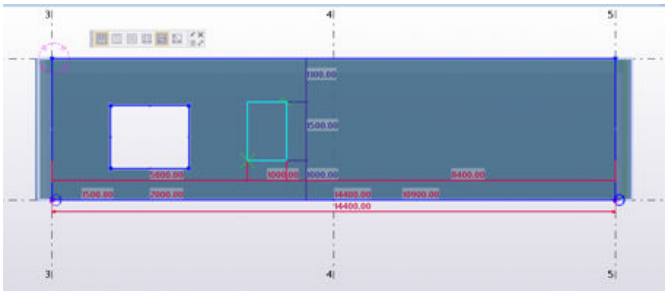
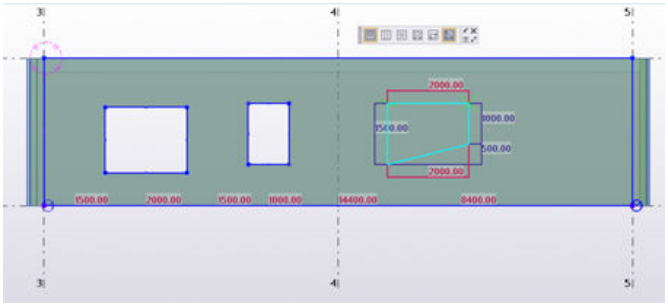
To add an opening, select the wall layout and then select the appropriate direct modification command. You can create rectangular, polygonal and circular openings.

NOTE You can use the **Wall layout opening** tool from the **Applications & components** catalog to define the settings applied to the openings. You can define the opening dimensions and select to apply detailing by using an opening component.

Add an opening to a wall layout

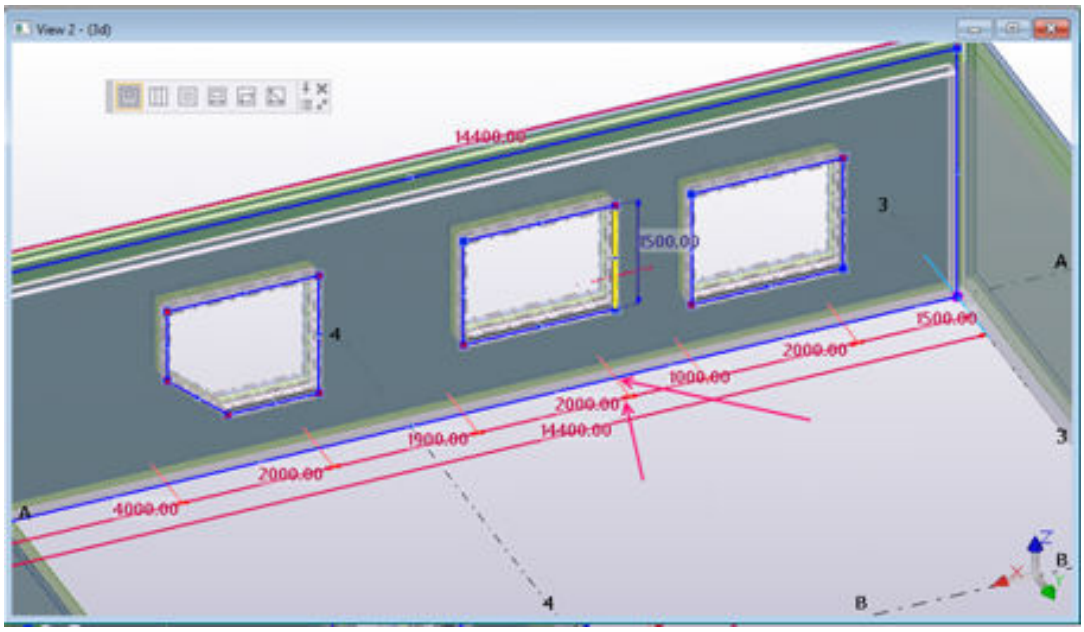
Option	Description
	Add an opening by picking the opening length in the wall. There are five different dimension options to define the opening bottom and top level. To cycle through the options, click the

Option	Description
	opening dimension button until it shows the wanted option. The five options are: • Enter the distance between the bottom of the wall and the bottom of the opening, `800.00` in this example, and the height of the opening, `2400.00`. If **Position in depth** values are used, the bottom distance is measured from the creation points. • Enter the distance between the bottom of the wall and the bottom of the opening, and the distance between the bottom of the wall and the top of the opening. If **Position in depth** values are used, the bottom distance is measured from the creation points. • Enter the global Z coordinate of the opening bottom level, and the height of the opening. • Enter the height of the opening, and the global Z coordinate of the opening top level. • Enter the global Z coordinate of the opening bottom level, and the global Z coordinate of the opening top level. You can use an architectural drawing as a reference model where you can pick the opening width and the location. You can select the layer to be cut on the contextual toolbar. This makes it easier to model different layer geometries to walls with multiple layers. The default option **All layers** creates the opening to all layers. To change the opening creation for a layer, click the **Modify wall boundary offsets** command. Select the layer and either **Create** or **Do not create**.

Option	Description
	Add a rectangular opening by picking two points. 
	Add a polygonal opening by picking three or more points. 
	Add a circular opening by picking the center point and the radius point for the opening. When you change the layer and edge specific offsets, you can set a single offset for the circular opening. Circular openings enable fast creation of HVAC reservations.

Modify an opening in a wall layout

Modify the geometry of an opening

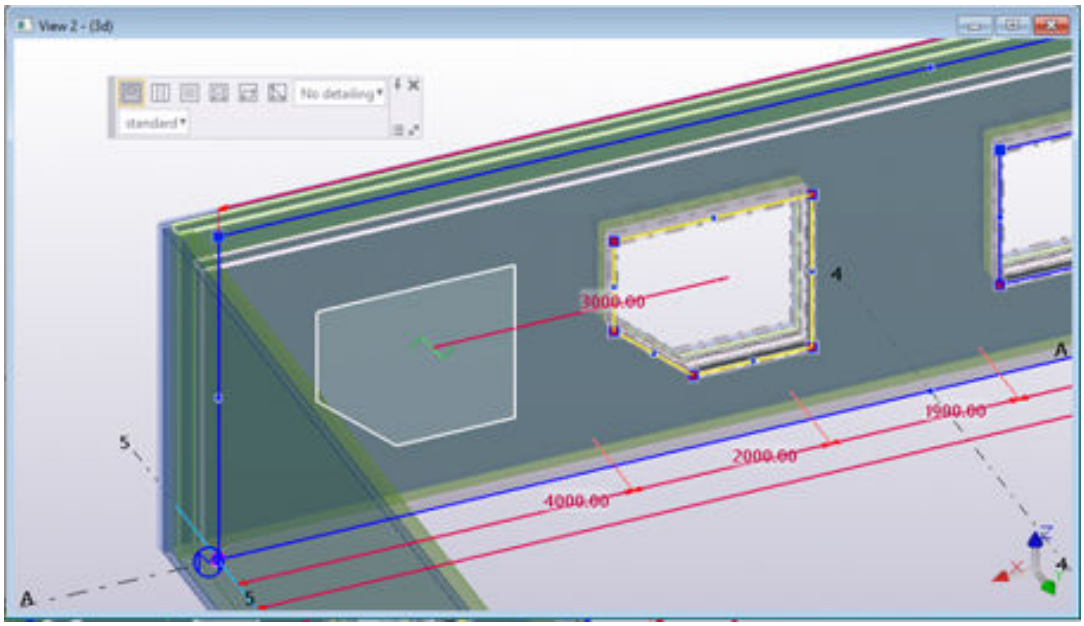


You can modify an opening in the same way as the outer edges of the wall layout by using the appropriate direct modification command. You can:

- Drag the edges.
- Drag the corner handles.
- Insert vertices by dragging the edge midpoint handles.
- Delete vertices by selecting the corner handle or edge, and pressing the **Delete** key.

In addition, you can modify the start or the end of the opening by dragging the line handles, or by using the dimension arrowheads.

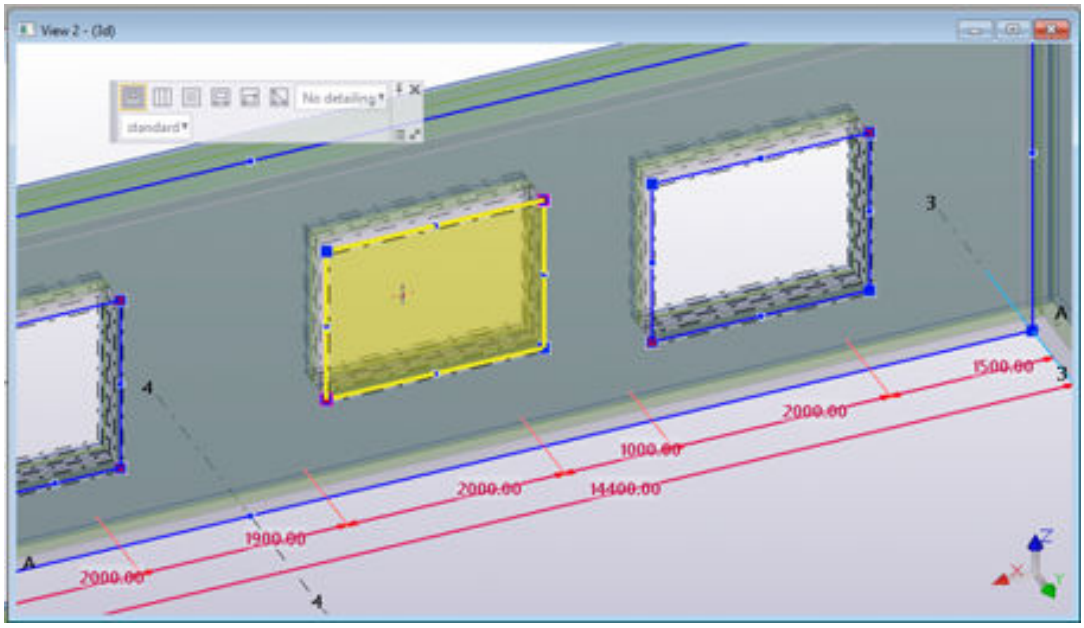
Move and copy an opening



1. Select an opening by selecting a face inside the opening.
2. Start dragging the selected face.
3. To create a copy of the opening, hold down the **Ctrl** key.

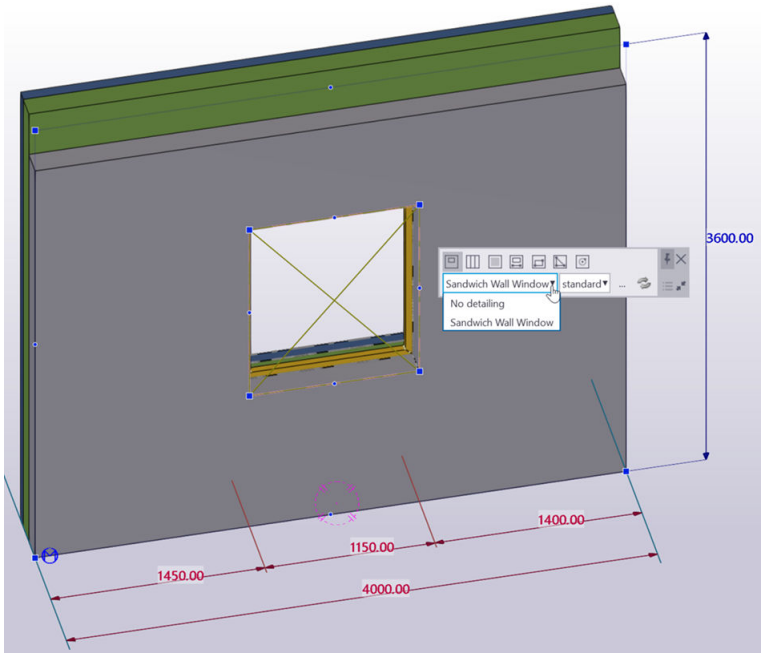
   To move an opening without changing the shape of the opening, drag the face handle to the desired location.

Delete an opening



Select a face inside the opening and press the **Delete** key.

Modify the opening detailing



You can create an opening with or without details. To detail an opening, select the window and select the predefined detailing component from the list. Note that you can use the **Sandwich wall window** component for rectangular openings, and for openings that connect a window and a door with the same top edge line.

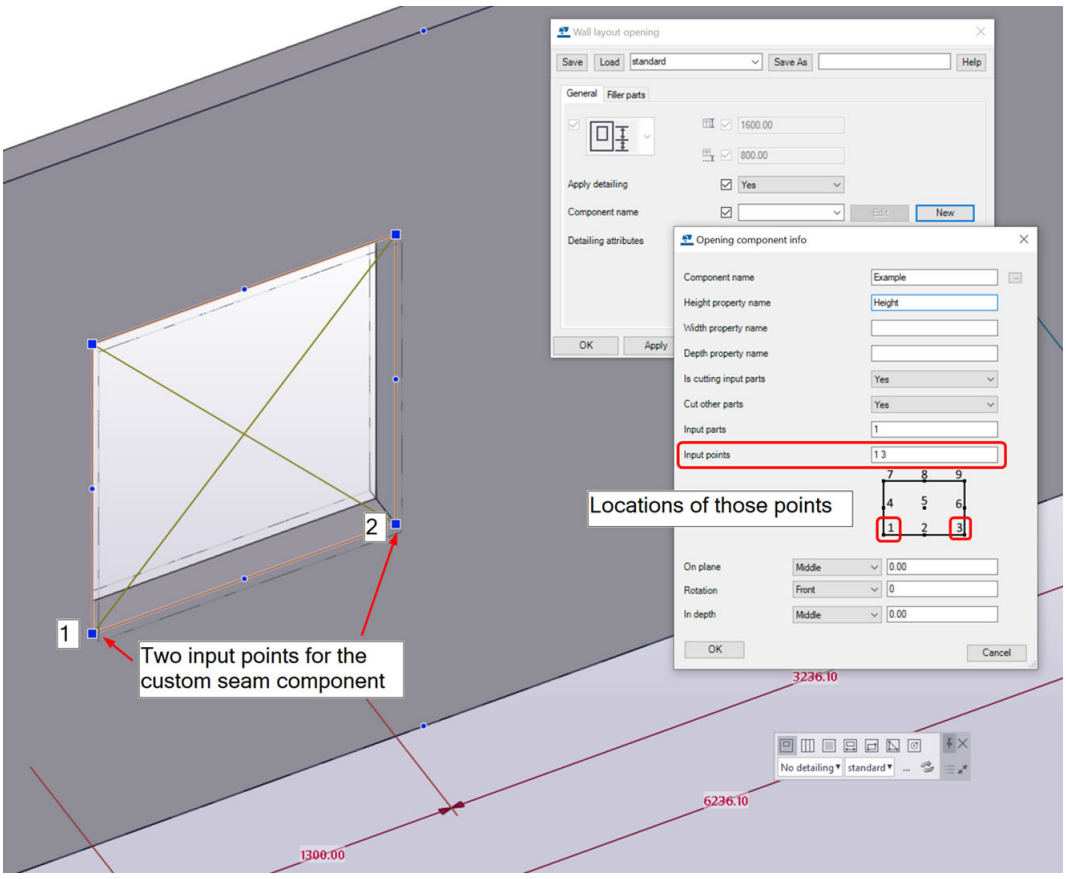
To define new detailing components to the list, select the opening and double-click to open the **Wall layout opening** dialog box or click the button with the three dots in the contextual toolbar.

1. In **Apply detailing**, select **Yes**.
2. In **Component name**, do one of the following:
   - Select a component from the list. Click **Edit** if you want to modify some of the opening properties.

     If you edit the properties and give a new name to a component, the edited component is added to the **Component name** list.
   - If you want to use a component that is not in the list, click **New** to select a component from the **Applications & components** catalog and define the opening component properties. You can use a custom component that you have created, for example.

     In the **Opening component info** dialog box, define the following:

Component name	Select the sub-component for the opening detailing.

Height property name	Name of the property that defines the height in the component (for custom components, P1, for example).
Width property name	Name of the property that defines the width in the component (for custom components, P2, for example).
Depth property name	Name of the property that defines the depth in the component (for custom components, P3, for example).
Note that all of the dimensions may not be needed in the property name. If you use a custom seam type custom component, we recommend that you have two input points to define the width of the opening. When you bind the opening details to the component input points in the custom component editor, you do not need to define the **Width property name**. In the example image below, the bottom corner points are mapped to the opening corners, setting the values `1 3` to map the location.  	
Is cutting input parts	Controls the input parts of the seam. • **No**: The detailing component does not create the cut, instead, **Wall**

	layout will create a rectangular opening cut. • **Yes**: **Wall layout** does not create a rectangular opening cut.
Cut other parts	Use for other layer parts (if any) which are not input parts for the seam component. This property is used when the wall has more than one layer. • **Yes**: **Wall layout** creates the cut. • **No**: **Wall layout** does not create a separate cut for other than input parts (if any).
Input parts	Input parts expected by the sub-component. If the sub-component is built to create details for a sandwich wall, for example, and it expects three parts as input, type `1 2 3`. Those are the layers and order defined for the wall type in **Wall layout**.
Input points	If the sub-component has two input points for defining the size of the opening, those points are mapped here to the opening corners. See the example image above for how to map the bottom corner points.
On plane **Rotation** **In depth**	On plane, rotation and depth properties only apply to components that have position parameters (custom part).

You can also add filler parts to the openings on the **Filler parts** tab. Select a layer in the example image in the dialog box. Then select whether the layer is cut and a filler part created to the layer. Select the appropriate filler part attributes. Note that the filler parts use contour plate properties in **Filler part attributes**.

You can also activate detailing on the contextual toolbar by selecting the detailing component that you want to use.

Wall layout seam

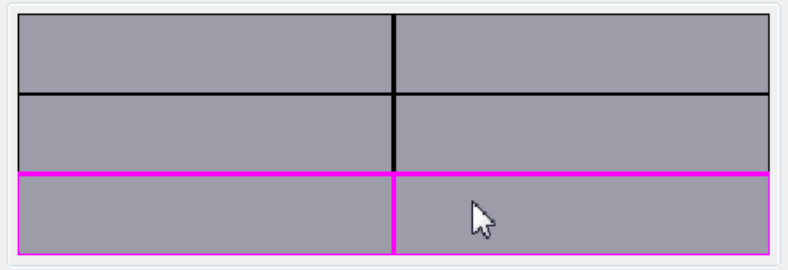
When you create a wall layout, it is at first a single precast wall. **Wall layout seam** divides the wall layout into two or more precast wall pieces. In the wall

layout, the wall layout layers that have the **Layer elementation** option set to **Yes** in the **Wall layout** dialog box are divided.

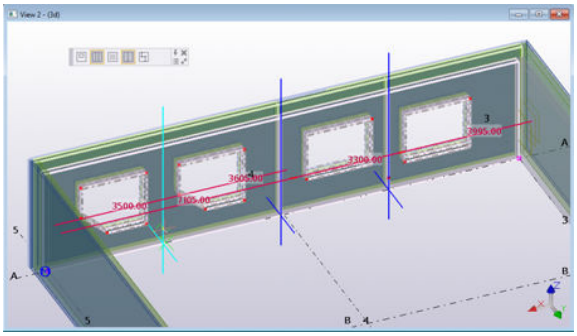
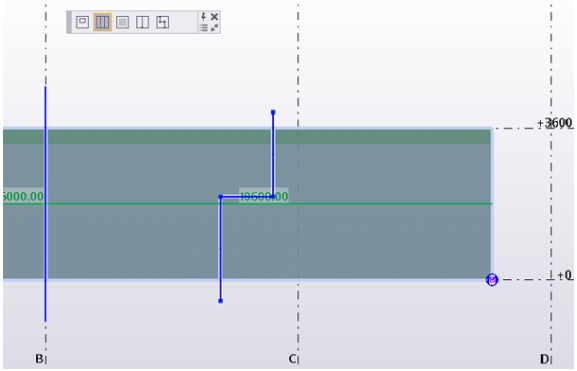
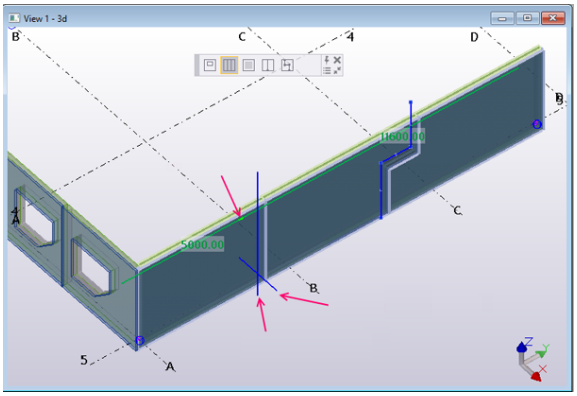
Add and modify wall layout seams

Use the **Modify seams** command on the contextual toolbar to add and modify seams. **Modify seams** opens the **Wall layout seam** dialog box where you can define the seam properties for each wall layer separately.

Select the layer in the preview image and define the properties.



Option	Description
	1. Define the gap dimension at the seam. 2. Define the gap offset from the seam input position.
Create a connection component	Select **Yes** to create a connection between the two layer parts at the seam. When you select **Yes**, you can define the other component options.
Component name	Select a connection component from the **Applications & components** catalog.
Component attributes	Select the attribute file for the connection.
Component primary input	Select which wall piece is the first input part for the connection, seam, or other components created at the seam.

Option	Description
Add a vertical seam 	Use the **New seam** command to add a vertical seam. Pick a point to create the seam. Note that **New seam** is selected automatically when you use the **Modify seams** command and the wall layout does not have any seams yet.
Add a polyseam 	Use the **New polyseam** command to add a polyseam. Pick two or more points to create the seam. Polyseams can contain vertical, horizontal, and sloping segments. Extend the start and the end of the polyseam to make it clearly cross the edges of the wall it is dividing.
Move a seam 	Move a vertical seam by dragging the seam. You can also move vertical seams by dragging the horizontal line handle or the dimension arrowhead. Move and modify a polyseam like any other polygonal object.
Copy a seam	Copy a vertical seam or a polyseam by holding down the **Ctrl** key and dragging the seam.
Delete a seam	Delete a vertical seam by selecting the seam handle and pressing the **Delete** key. Delete a polyseam by deleting all segments of the polyseam.

Option	Description
Modify the seam properties	1. Select a vertical seam or a segment of a polyseam. When you select the first seam, the current properties of the seam are loaded into the **Wall layout seam** dialog box. 2. Select two or more seams. Hold down the **Shift** key and then select the vertical seams, or the segments of a polyseam. 3. When you have selected the seams, modify the properties in the dialog box and click **Modify**.
Split the wall layout at selected seam	Use the **Split the wall layout at selected seam** command to split the wall layout into separate components.

Wall layout connector

Use **Wall layout connector** to apply a connection between two wall layouts. **Wall layout connector** fits the layer parts at the corners. You can control the corner gaps by extending or shortening the layers of the walls, and by adding connections between the connected layer parts. **Wall layout connector** can also add connections between the layer parts when you are using a connection component at the corners.

Note that if you have created several wall layouts at one go using the **Wall layout** tool, wall layout connectors are automatically added between the wall layouts. Wall layout connector is updated automatically when you change the wall type.

If you create a wall layout and later add another next to it, you can add a connection between them using the **Wall layout connector** component in the **Applications & components** catalog. Select the two wall layouts to create the connection. Ensure that you select the wall layout component instance, not a single object created by the wall layout. The connector is created when you select the second wall layout.

Wall layout connector properties

Option	Description
	Define how the layer parts are fitted. Click the button three times to cycle through the three different corner set-up options.
	Define whether the panel ends are square or sloped. You can use this option to set a typical corner detail for double walls. This option is also applied in a connection component used between the layer parts.
	Define how the parts forming the turning corner are modeled. You can select this option separately at each layer. When you select this option, **Wall layout connector** creates an additional part that is added to the cast unit. The turning corner part is created with the first name given in the system file `SandwichWallCornerPartNames.dat`. If the file does not exist or if it is empty, the main part name is used. This way the turning corner area gets automatically calculated into the custom quantity `CUSTOM.WALL_CORNER_AREA`. Click the button three times to cycle through the three different corner set-up options.
Free space A	Define the gap between the two layer parts. The preview image shows where the gap is located. The gap location depends on the corner set-up. **TIP** If you are using a connection component, use property name `FreeSpace` in the

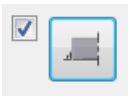
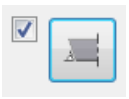
Option	Description
	custom connection to control that value from **Wall layout connector** directly.
End offset B	Define the end offset of the longer part. The preview image shows where the offset is located. The offset depends on the corner set-up. **TIP** If you are using a connection component, use property name `EndOffset` in the custom connection to control that value from **Wall layout connector** directly.
Connection	Select a connection component that is created between the two layer parts. If you do not select a connection, only fittings are added according to the gap and the end offsets. If you want to view the current connector properties, select the wall layout connector or double-click the fitting created by the connector.
Attributes	If you have added a connection, select an attribute file for it.
Connection main part	If you have added a connection, set the connection main part.

Wall layout T connector

Use **Wall layout T connector** to apply a T connection between two wall layouts. **Wall layout T connector** can create a seam at the connection location. Note that if you remove the connection, the seam is not automatically removed. Modify the main part to remove the seam.

Wall layout T connector properties

Option	Description
   	Define how the wall layout parts are fitted. Click the button three times to cycle through the three different corner set-up options. Select the wall layout in the preview image and define the connection properties.

Option	Description
 	Define whether the panel ends are square or sloped. This option is applied in a connection component used between the wall layout parts.
Add seam	Select whether to add a seam. You can also select to only split the first layer into two parts. **No** does not split the first layer part but creates a cut that breaks the part into two solid objects.
Seam offset	Enter the seam offset.
Free space	Define the gap between the wall layouts. The preview image shows where the gap is located. The gap location depends on the corner set-up.
Create component	Select whether to use a component to connect the wall layouts.
Component name	Select a connection component that is created between the wall layouts.
Component attributes	If you have added a connection, select an attribute file for it.
Component primary input	If you have added a connection, set the connection main part.

Wall layout elementation

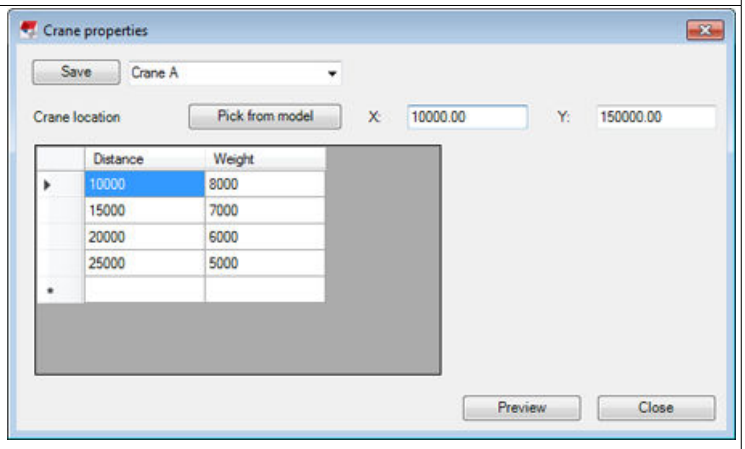
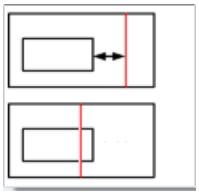
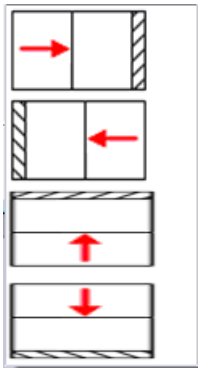
Wall layout elementation automatically adds new seams to divide the wall layout into two or more precast wall pieces.

Openings are taken into account when adding the seams. Openings may affect the final seam position so that the final length or weight of the precast wall is less than the target value.

NOTE Weight calculations are based on the wall layout geometry. Any other additional parts in the walls are not taken into account in the calculation. The density in weight calculation is 2500 kg/m³ for the structural layer type and 100 kg/m³ for the insulation layer.

Option	Description
Create seams	• **By length**: Enter one or more desired lengths. If it is not possible divide the wall using the first length, **Wall layout elementation** tries to use the next length you have entered, and so on.

Option	Description
	• **By number**: Enter the number of walls you want to create. • **By weight**: Enter the desired weight of a wall piece. The wall layout is divided so that the weight of the precast walls is as close to the desired weight as possible. • **By crane lifting weight**: Divides the wall layout so that the weight of the precast walls is as close to the maximum lifting capacity as possible.
Max height	Enter the maximum height of a wall piece. If the value you enter is smaller than the height of the wall, **Wall layout elementation** will first create the horizontal seams with the given height and settings. After that it creates the vertical seams using the vertical seam settings and all other settings in the dialog box.
Min length	Enter the minimum length of a wall piece.
Max length	Enter the maximum length of a wall piece. This value is used with all creation methods.
Max weight	Enter the maximum weight of a wall piece. This value is used with all creation methods.
Max crane lifting weight	Select the crane used for erection. When the crane is selected, the maximum weight of the walls is always less than the lifting capacity of the crane. To define a crane set-up, open the **Crane properties** dialog box by clicking the … button on the right: 1. Pick the crane location in the model, or enter the X and Y values. 2. Enter the crane lifting capacity data in the table. Enter the distance in the from-near-to-far order. 3. Save the crane set-up. You can use the crane name when saving the set-up. To modify and review the existing crane set-up, select the crane set-up from the list in the **Crane properties** dialog box. Click **Preview** to see the crane location on the lifting graph visualized in the model view.

Option	Description
	
Avoid openings 	Select whether the seams may cross the opening. If the seams may not cross the opening, enter the minimum distance from the opening.
Distribution direction 	Select the direction of the seams, and the start end at the wall layout. You can add both vertical and horizontal seams.
Vertical seam settings **Horizontal seam settings**	Select the seam attributes used for new seams. Ensure that the layer offsets are set properly as they may affect the weight and length of the wall.
Create to selected walls	Select the wall layout and click the **Create to selected walls** button to insert the seams. You can select one or multiple wall layouts.

Wall layout layer swapper

Use **Wall layout layer swapper** to swap the structural precast layers of one single wall in a wall layout, typically a double wall. You can swap the order of

some individual walls in the same wall line if they need to be cast in a different order.

To swap layers, select **Wall layout layer swapper** in the **Applications & components** catalog, then select a wall layout and the wall part that you want to swap.

When you use the swapper tool, a triangular handle is shown at each location of the tool . You can drag these handles to modify the wall to be swapped. To swap another wall in the layout, you can copy the swapper to another location by holding down the **Ctrl** key while dragging.

You can delete the swapper instances by selecting the swapper handles and pressing the **Delete** key.

Customize the wall layout

UDA tab content

You can customize the content of the **UDA** tab by using the `WallLayout.Udas.dat` file. The `WallLayout.Udas.dat` file can be located in any of the folders set in the `XS_FIRM`, `XS_PROJECT` or `XS_SYSTEM` advanced options, or in the model folder.

You can use the file to control which user-defined properties of the created layer parts can be used in the wall layout. We recommend that you only use UDAs that are typically common for all precast walls in one wall layout.

You can add the `tab`, `label`, and `endlabel` data types in the file to organize the content of the **UDA** tab according to specific workflows.

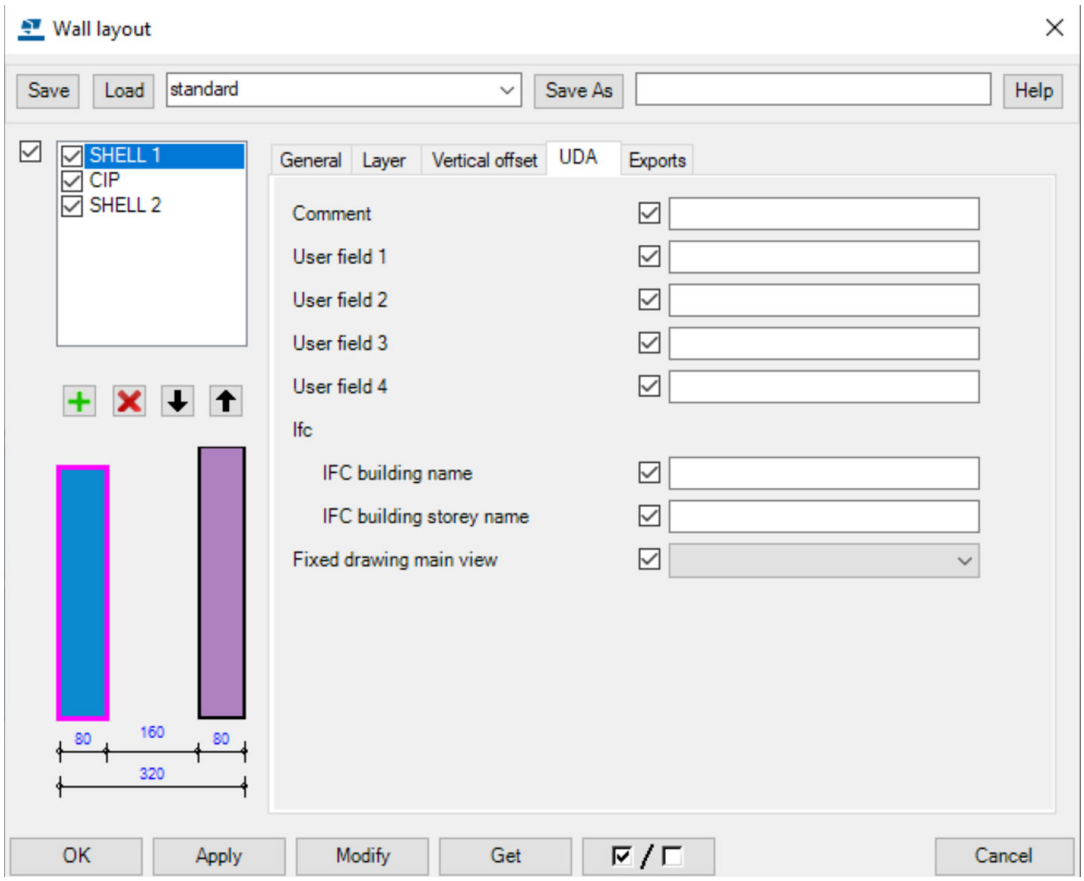
You can define that layer UDA values are not swapped when the **Wall layout layer swapper** tool is used. Add `NoSwap` at the end of the line (fourth field): `string comment j_comment NoSwap`.

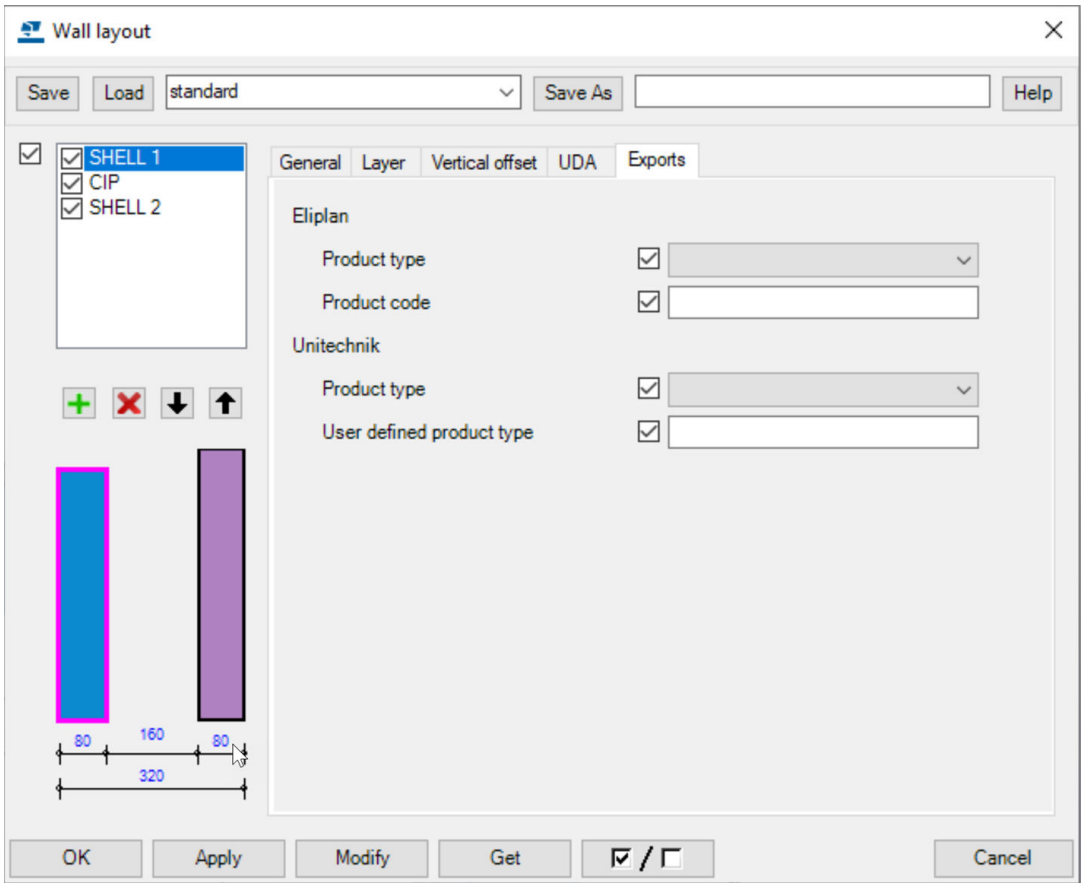
The sample file shown below contains a full description of all the settings and the format of the file. The lines starting with '//' are comment lines.

```
// Customized user defined attributes (UDA) for layer parts created by
wall layout component
//
// Each row shall contain 2 or 3 fields separated by tab(s) or semicolon.
// Please note that all uda names shall be unique
//
// Field 1: The data type of the attribute. Valid values are 'distance',
'float', 'option', 'integer', 'string', 'tab'*, and 'label'*
//          'integer' and 'string' compatible with the actual user
defined attributes as
//          specified in objects.inp
// Field 2: UDA name. This is the name of the user defined attribute set
for the layer part.
// Field 3: Label. This text is shown in the attributes dialog. It can be
a translatable label (albl_) or any text.
//        leaving this field empty will use the orignal translation
strings.
// Field 4: 'NoSwap' if you do not want the 'Wall layout layer swapper'
to swap the values.
```

```
//
// *Attribute 'tab' will place all the UDAs below to a new tab. In field
2 the name of the tab can be set.
// *Attribute 'label' will create a valueless label and will indent the
UDAs below. In field 2 the name of the label can be set.
//             Optional: to close the label use 'endlabel' to return to
unindented UDAS.
string         comment             j_comment
string         USER_FIELD_1
string         USER_FIELD_2
string         USER_FIELD_3
string         USER_FIELD_4
label         Ifc
string         IFC_BUILDING         j_IFC_building
string         IFC_BUILDING_STOREY     j_IFC_building_storey
endlabel
option         FixedMainView         j_FixedDrawingMainView
// new tab
tab         Exports
label         Eliplan
option         EP_TYPE             j_Product_type
string         EP_CODE             j_Product_code
label         Unitechnik
option         UT_product_type         j_UT_product_type
integer         UT_UD_product_type     j_user_defined_product_type
```

The images below show how the settings defined in the example file are created in the **Wall layout** dialog box.





Layer component

If the custom part or component property names are not the same as the default names, you can define the parameters in the `xxxxx.LayerComponentInfo.xml` file where `xxxx` is the component name.

In this file you can also specify the mapping for other layer properties so that the values given in the attribute file will be overridden with the layer properties defined in the **Wall layout** dialog box.

An example of the file is shown below.

```
<?xml version="1.0" encoding="utf-8"?>
<LayerComponentInfo xmlns:xsd="http://www.w3.org/2001/XMLSchema"
xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance">
  <ComponentName>CMUWall</ComponentName>
  <!--Leave the property name empty if you don't want anything to be set
by Wall layout -->
  <HeightMaxProperty>WallHeight</HeightMaxProperty>
  <Height1Property></Height1Property>
  <Height2Property></Height2Property>
  <ThicknessProperty></ThicknessProperty>
  <NameProperty>Name</NameProperty>
  <ClassProperty></ClassProperty>
  <MaterialProperty>Material</MaterialProperty>
  <PrefixProperty>AsmPfx</PrefixProperty>
  <StartNoProperty>AsmStNo</StartNoProperty>
  <!--Do not specify anything for custom part position if you want to
control these with saved settings only.-->
```

```
  <CustomPartPosition>
    <PlaneOffset>0</PlaneOffset>
    <DepthOffset>0</DepthOffset>
    <RotationOffset>0</RotationOffset>
    <Plane>LEFT</Plane>
    <Depth>BEHIND</Depth>
    <Rotation>FRONT</Rotation>
  </CustomPartPosition>
</LayerComponentInfo>
```

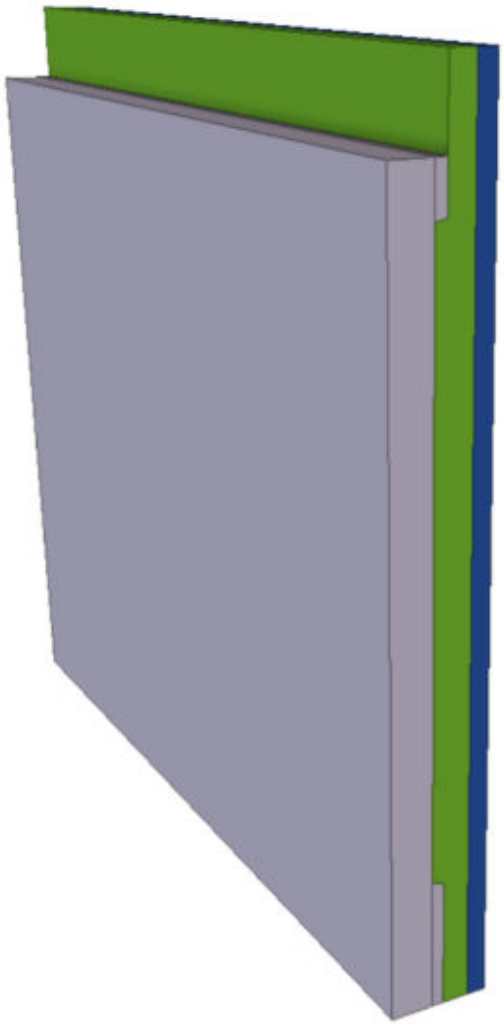
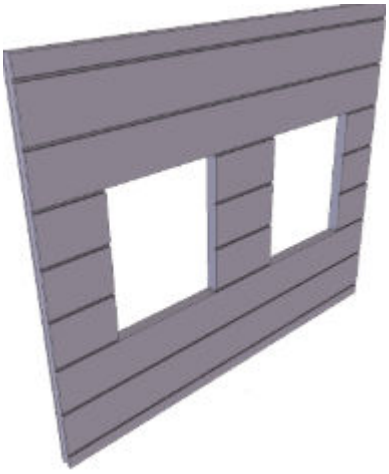
Geometry detailing strip

Geometry detailing strip allows modeling of wall edge thickenings, recess patterns, wall corbels, mock joints, and facade protrusions. Note that this component is for precast use only.

Geometry detailing strip can be used for adding detailed geometry to any object. You can create different settings for **Geometry detailing strip** and select the appropriate settings for each situation.

The tool uses irregular concrete profile catalog profiles. If these profiles are not found in your Tekla Structures environment, download them to your model from the Tekla Warehouse local content.

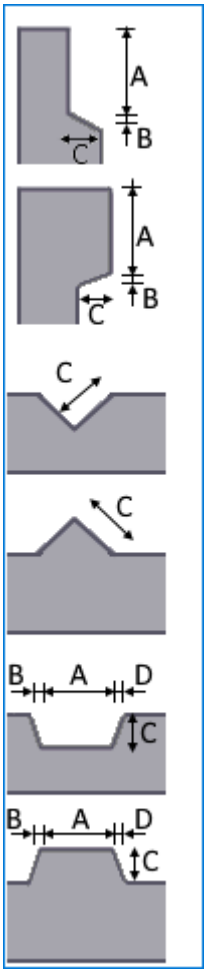
Use for

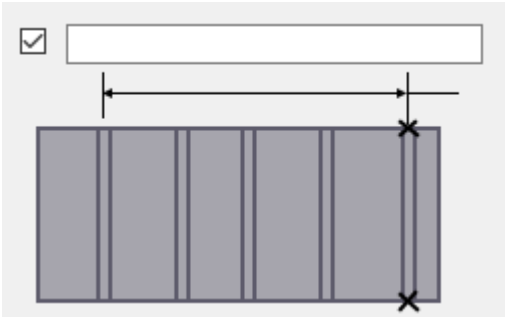
	Sandwich wall thickening
	Recess patterns in a wall

Selection order

1. Select the part face.
2. Pick two or more points to indicate the location.
3. Click the middle mouse button to create the geometry strip.

Geometry detailing strip properties

Option	Description
	Select the geometry. You can create thickenings, recesses, and protrusions. Define the A, B, C and D dimensions for the selected option.
Add to cast unit, **Attach to part**	Select how to add the geometry to the selected part: Attach to part or Add to cast unit.
Cut only, **Cut and part**, **Part only**	Select how the cut is modeled.
Use main part properties	You can control the properties of the created part. With the **Use main part properties** option, the added part gets the properties of the main part. Otherwise, you can select the saved

Option	Description
	standard attributes for the concrete beam. Note that if you change the saved properties afterwards, you need to modify the geometry detailing strip to get the properties of the added part updated.
Middle. **Left**, **Right**	Select a position for the geometry.
	Define the strip offset from the input line.
	Define the spacing of recesses and protrusions. For example, enter 40*400 to create 40 recesses with a spacing of 400, measured from the center of the recess.

Formwork placing tools

This section introduces formwork placing tools.

Click the links below to find out more:

- Formwork placing tools - Walls
- Formwork placing tools - Walls: configuration
- Formwork placing tools - Slabs (page 3645)
- Formwork placing tools - Slabs: configuration (page 3653)

Formwork placing tools - Walls

Formwork placing tools - Walls is a set of components that helps in detailed modeling of different wall panel formwork systems. These components are placing tools, and therefore you need to have all relevant formwork products such as wall panels, corner conditions, clamps, and pouring platforms available in the **Applications & components** catalog.

You can get the formwork products provided by formwork suppliers from **Tekla Warehouse**, for example. In addition, **Formwork placing tools - Walls** requires additional configuration files that contain the necessary information about the formwork product components. These configuration files may come

with the formwork product catalogs provided by formwork suppliers, but you can create the configuration files yourself as well.

Formwork placing tools - Walls contains tools to place and modify the following formwork elements:

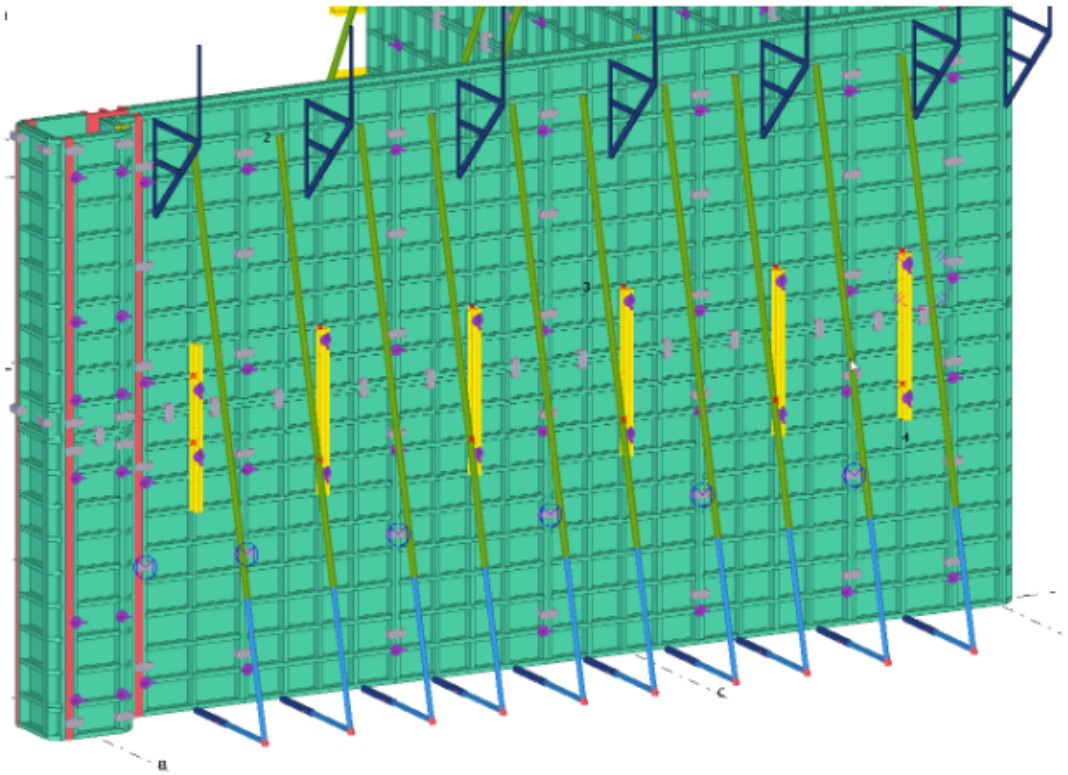
- formwork wall panels typically appearing as two-dimensional arrays covering a certain area
- different kinds of corner conditions
- an array of ties between two formwork panels
- an array of clamps, locks, or brackets at the joining edge between two formwork panels to keep the formwork together
- supporting braces for a single formwork panel
- an array of pouring platforms at the top edge of formwork panels
- fillers to fill the gaps between two formwork panels

Because **Formwork placing tools - Walls** consists of a set of sub-tools which are combined in one dialog box, each tab is an individual sub-tool. Both the **Formwork placing tools - Walls** and the sub-tools are available in **Applications and components** catalog.

Most of the settings in the components are preset. You can control the different settings by selecting a suitable options in the dialog box. These preset settings are organized according to the formwork supplier and the product families.

However, if you create your own configuration files, you can use the values in the configuration files instead of the preset values.

You can access the **Formwork placing tools - Walls** in the **Concrete Contractor**, **General Contractor** and **Rebar Detailer** roles in the **Default** environment.



Panel tab

Use the **Panel** tab to model an array of formwork panels on one side or on two sides of a cast-in-place wall.

The actual formwork panels are custom parts. In addition to the custom parts, some additional data of the products is needed from the configuration files.

1. On the **Panel** tab, in the **Formwork supplier** and **Product family** list, select the suitable options.

   The options vary depending on which catalogs and tool configurations you have in use.

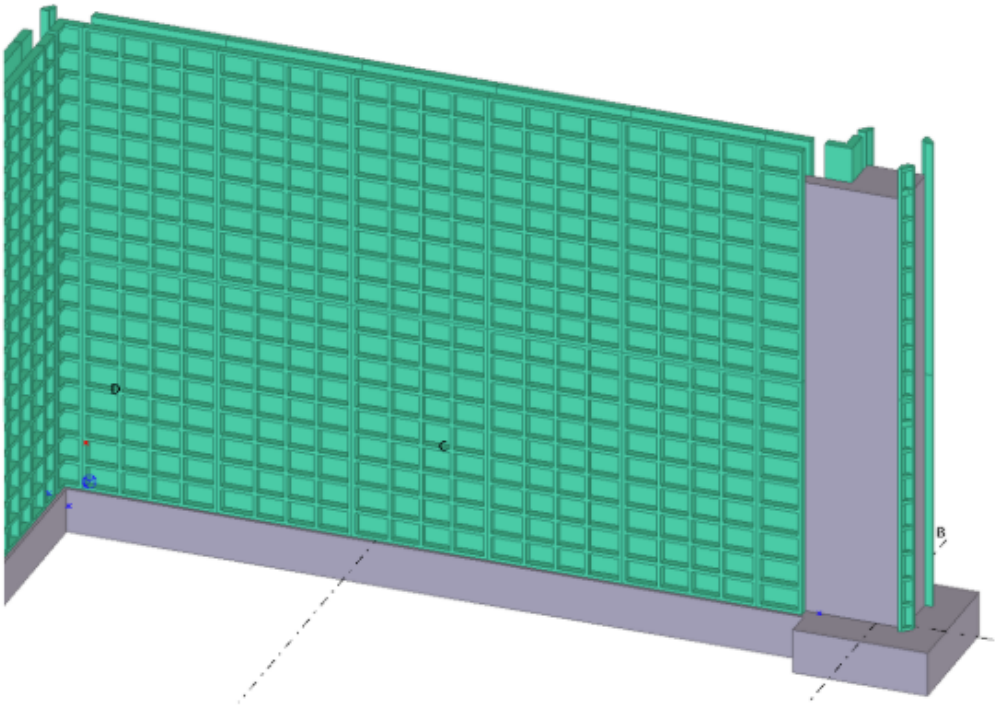
2. Select the panel size in the **Formwork panel** list, and set the other panel properties such as the layout, height, and fill location.

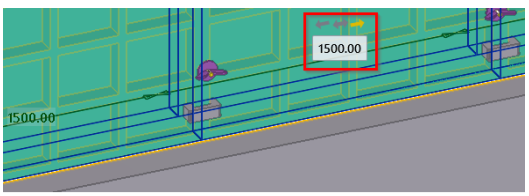
3. Click **Apply** and **Insert new** to start placing the wall panels.

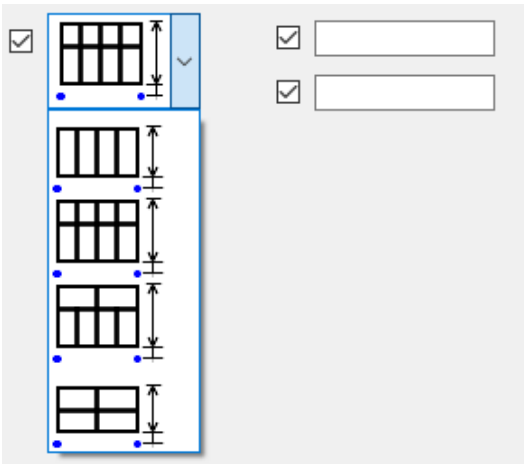
   Move the mouse pointer over pour objects or concrete parts to select a suitable location.

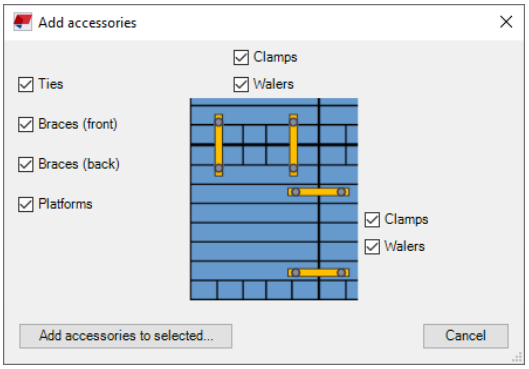
4. Pick the start point and the end point of the formwork wall panel.

   A preview of what the panels would look like after being placed is shown and you can decide which panels should be created. You can also change the panel layout afterwards by using direct modification.



Option	Description
Formwork supplier **Product family**	Select the formwork supplier and the product family.
Formwork panel	Select the panel size. If the total length of the panel array does not match with the panel width, smaller panels with the same height are automatically placed in the array near the fill location (start/middle/end of the array).
Avoid panels	Define which panels are avoided during the automatic insertion.
Panel layout	Override the default panel widths. When you insert the array for the first time, leave the value empty and modify the panel widths by using direct modification. 

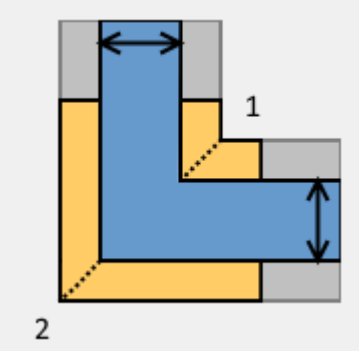
Option	Description
	Select the layout of the panel array. Enter the offset from the wall bottom and the total height of the panel array.
One side **Two sides**	Select whether the panels are created on one side or on both sides of the wall. You can modify the wall thickness. When the walls are inserted, the applied wall thickness is overridden with the actual wall thickness.
Fill location	Use the **Fill location** option to control where the empty space for the filler is located.
Add to pour unit	Select whether the panels and all accessories related to the panels are added to the pour or not.
Select new pour...	The initial pour is identified automatically when the panels are inserted. To change the selected pour, click the **Select new pour...** button and select a new pour. Note that there is no automatic recognition of a new pour if you decide to move the formwork panels, or if you add pour breaks which change the pours.
Insert new	Click the **Insert new** button to start inserting new formwork panel arrays.

Option	Description
Add accessories to selected	Add various accessories automatically to the selected panels.  Select the check boxes for the accessories you want to add. Note that for clamps and walers you can control separately the creation at horizontal and/or vertical edges.

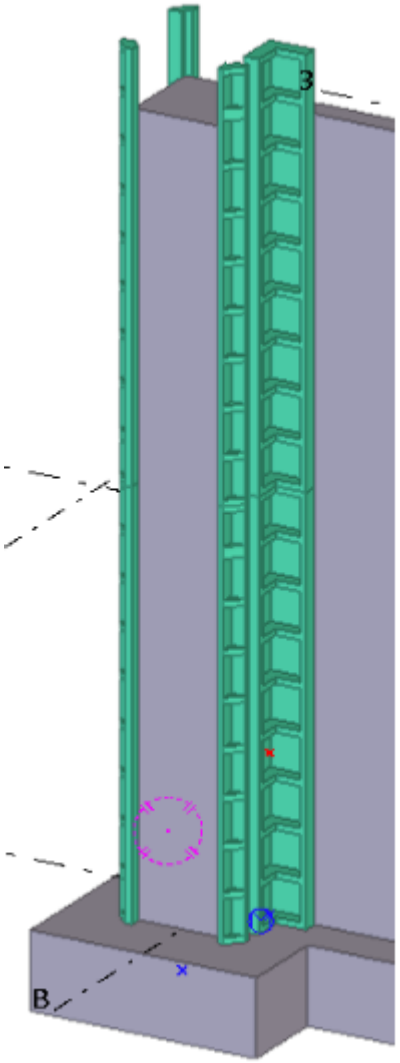
Conditions tab

A condition is a set of formwork components, such as custom parts, parts or other items, designed for special geometries in the wall. Conditions create the formwork at special locations in the walls, such as L, T, and X corners, bulkhead at the end of a wall, or pilasters at pour ends or at the middle of the wall. Each different type of a special geometry is its own condition type.

Each condition consists of two or more sub-assemblies of the formwork components, depending on the condition type. For example, the L corner contains the inner corner assembly **(1)** and the external corner assembly **(2)**. When inserted in the model, the location of these assemblies is controlled with the main insertion points and the two wall thickness parameters.



Use the **Conditions** tab to create a condition that consists of two or more sub-assemblies of formwork components.

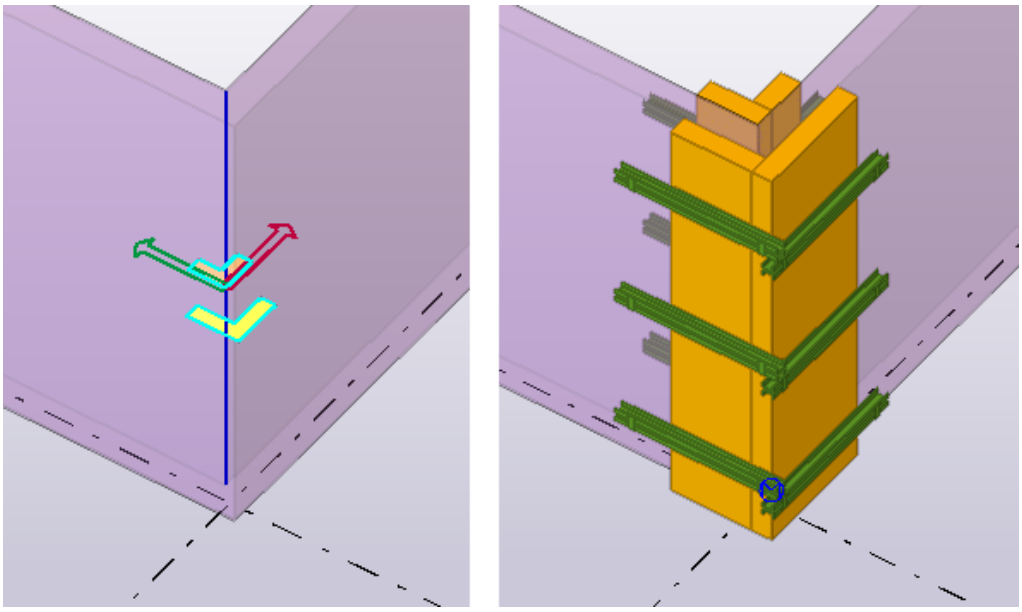


1. On the **Conditions** tab, select what type of condition to use by clicking the condition icon, and set the values to be applied for the selected condition.
2. Click **Apply** and **Insert new** to start placing the conditions.
3. Move the mouse pointer on the corner edges, pilaster edges, or ends of the pours or parts.

   Formwork placing tools - Walls automatically recognizes the condition type and wall thickness properties.

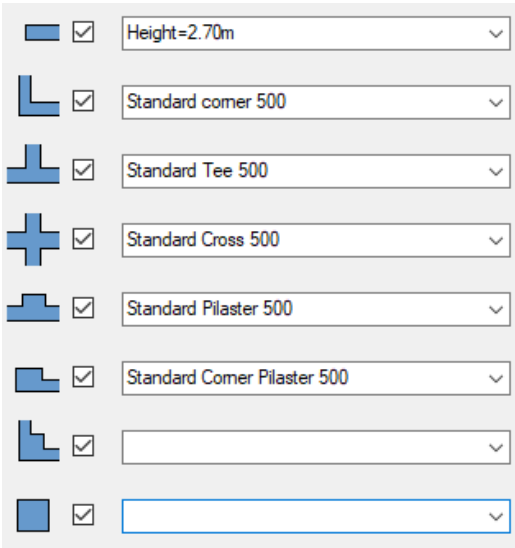
   Two arrows and a preview of the condition parts to be created is shown. The red arrow indicates the first direction and the green arrow indicates the second direction.

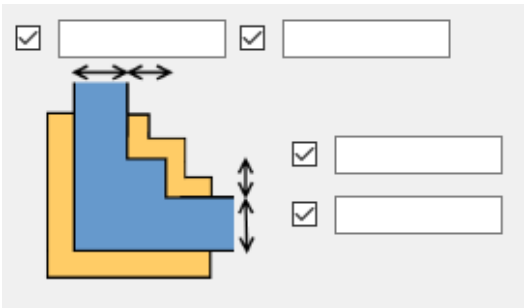
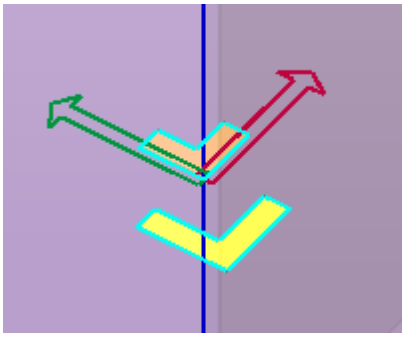
   In the example below, the longer panel becomes parallel to the red arrow, and the shorter panel will be parallel to the green arrow. Depending on the case, you may switch the arrows by moving the mouse on the other side of the edge.

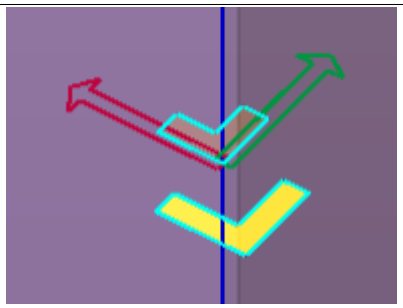


4. Click to place the conditions.

In some cases the condition tool may not recognize the condition type correctly. For example, if you want to insert an L corner to a location where the other side of the wall is very short it may happen that the condition tool is trying to insert a corner pilaster.

Option	Description
	Select the condition to be created. The selection is specific for each type of condition. Click the condition icon to select the condition type to be used.

Option	Description
	Modify the dimensions that control the location of condition sub-items in relation to the insertion point. When you insert a new condition, these dimensions are overridden by the calculated values based on the actual insertion location and condition type.
Offset from bottom	Define the vertical offset for the condition from the bottom of the wall.
Number of stacked conditions	To insert two or more vertically stacked conditions, enter the number of stacked conditions.
Height	Height of one stack when the conditions are stacked.
Left/right corner	Select how the conditions are inserted if you use the left-hand side placement. If you select **Rotate**, the condition always switches the red/green axis to a right-hand side placement. If you select **Mirror**, the left-hand side placement is allowed. Right-hand side placement:  Left-hand side placement:

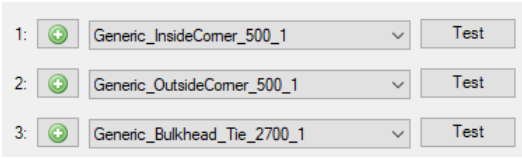
Option	Description
	
Filler configuration	Define which filler type is used when you place a condition which includes a filler.
Configure	Open the **Condition setup** dialog box to modify the existing conditions or to define new conditions.

Define a new condition

Defining a new condition has two phases. First, you need create the needed sub-assemblies. Once you have all the needed sub-assemblies available, you can define a new condition by giving it a name and defining which sub-assemblies are used in the new condition.

To define a new condition or to modify existing condition definitions, click the **Configure** button. The **Condition setup** dialog box opens where you can do the needed modifications.

Option	Description
Condition setup file	Conditions are saved in external data files (`xxxx.FormworkTools.Conditions.csv`). Select the file to add, modify or delete the defined conditions.
Save **Save as** **New**	Click **Save** or **Save as** to save any changes. Click **New** to create a new empty file.
Condition type	In the list, select the condition type to be modified. The list shows the existing conditions if there are any. If you have one or more existing conditions of the selected type, you can select the current condition in the list. When the condition is selected, the current

Option	Description
	definition values are shown and can be set or modified on the right-hand side of the dialog box. Click to create a new condition. Click to copy the selected condition. Click to delete the condition.
Formwork supplier **Product family** **Condition**	The condition has the supplier and the family properties. The conditions are available only when the supplier and the family are matching. The condition names need to be unique. Decide about naming conventions so that the names are as descriptive as possible.
	The condition can have 2-5 sub-conditions, depending on the condition type. One condition must have at least one sub-condition defined as otherwise it does not create anything. Click to open a sub-assembly wizard dialog box to create a new sub-assembly. Click the **Test** button to verify the output of the selected sub-assembly. To test the sub-assembly, pick a point in the model. The tool creates a sample wall and the sub-conditions with right side and left side placement. After you have done the testing, you can delete model objects that were created for testing purposes.
Total height for stacking	The total height of the condition can be calculated automatically based on the selected sub-conditions, or you can give the height. The total height is used as the spacing value when two or more conditions

Option	Description
	are stacked vertically. If the condition contains some parts sticking out and this part is overlapping in the stack, you may need to enter the total height manually.
Left/right corner	Test and verify how the **Mirror** and **Rotate** options affect the output of the condition.
Test condition	Test and verify the output of the whole condition, including all sub-conditions. To test the condition, pick a point in the model. The tool creates a sample wall and two conditions with right side and left side placement. After you have done the testing, you can delete the model objects that were created for testing purposes.

Create a new corner sub-assembly

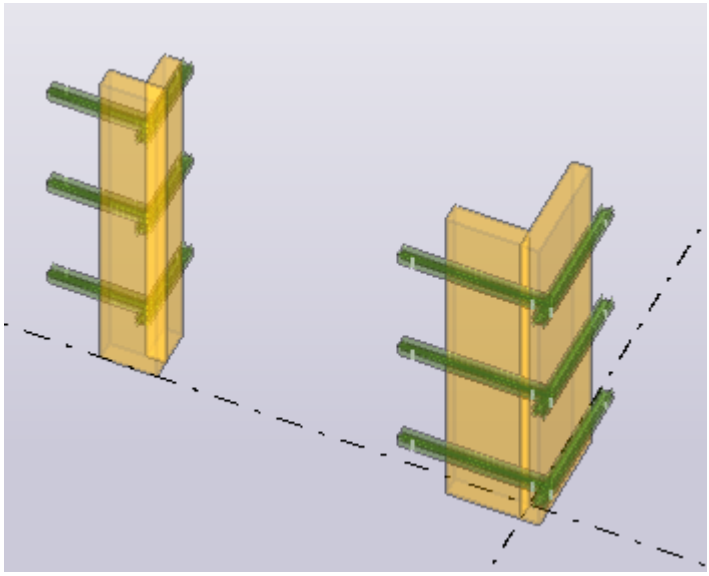
The corner sub-assemblies are created using **Formwork corner sub-assembly wizard**.

Typically, the corner sub-assembly contains formwork products at one side (inner or external) of the corner, or at some corner in a pilaster.

The outcome of the wizard is stored in an external file with a name.

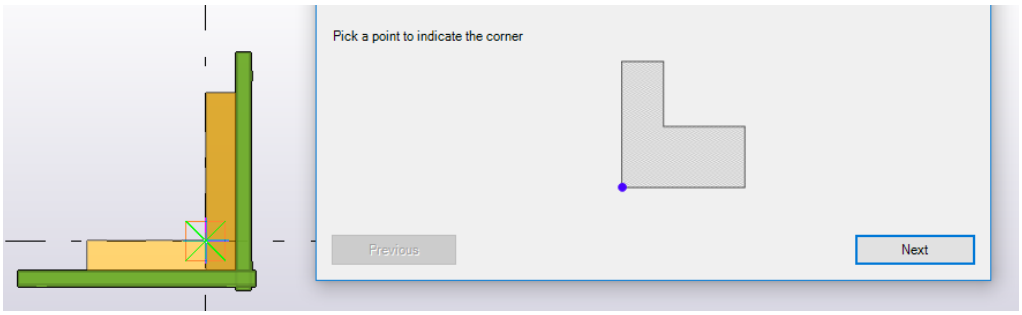
To open the dialog box, click in the **Condition setup** dialog box on a condition that requires a corner sub-assembly. Alternatively, you can search for the **Formwork corner sub-assembly wizard** in the **Applications and components** catalog.

Before you can start creating the corner sub-assemblies, you need to have an external or inner corner formwork in the model. When you have created the corner formwork, you can start the wizard.



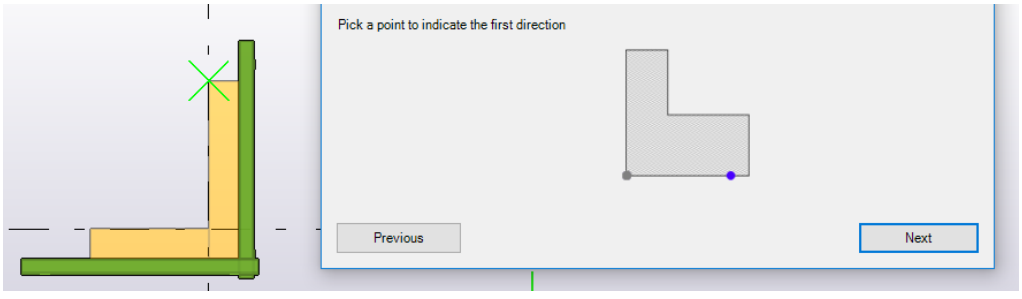
1. Pick the first corner point.

   This point will be placed exactly to the external or to the internal corner of the concrete in the wall in the final condition placement. Note that after you have picked the point, the wizard steps forward automatically. You can step back to re-pick a new point.



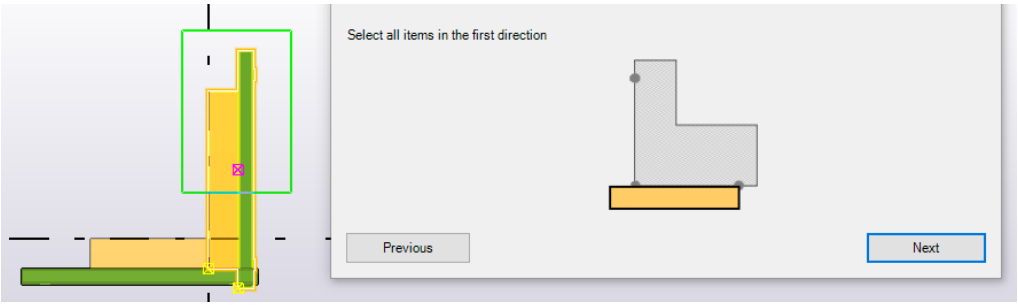
2. Pick a point showing the direction of the first wall.

   This is typically the red arrow in condition placing.



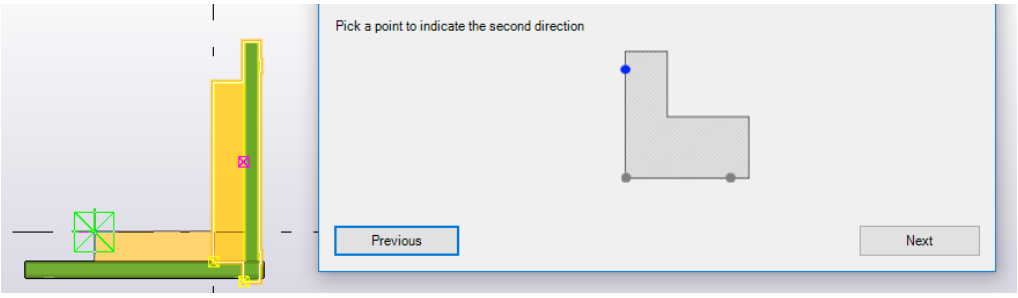
3. Select all formwork items belonging to the first wall.

   In practice, these formwork items will be parallel to the first wall when placing the sub-assembly of the condition.



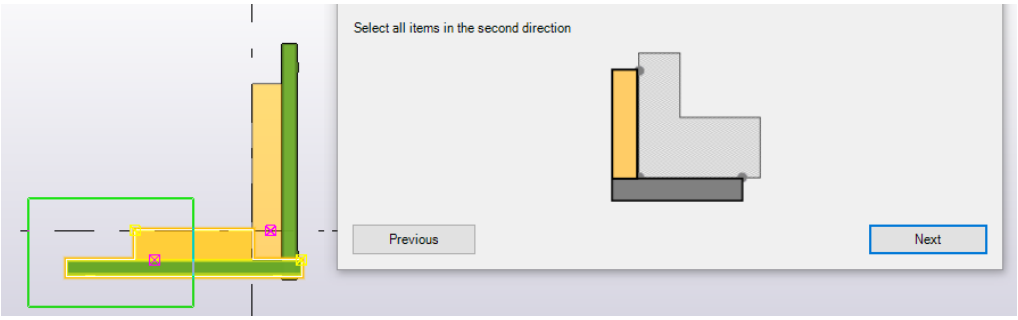
4. Pick a point showing the direction of the second wall.

   This is typically the green arrow in condition placing, and perpendicular to the first wall.

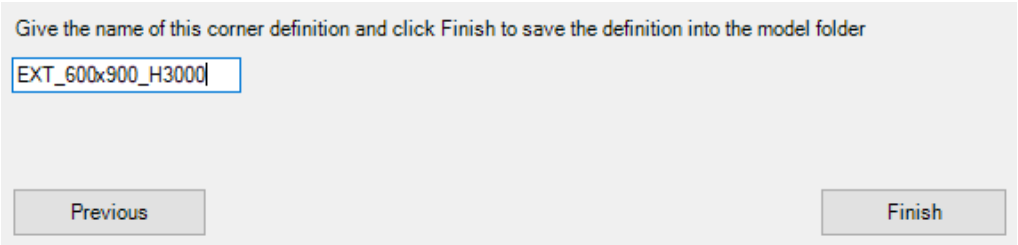


5. Select all formwork items belonging to the second wall.

   Typically, these formwork items will be parallel to the second wall when placing the sub-assembly of the condition.



6. Enter a file name to save the corner sub-assembly. Click **Finish** to close the wizard.



All corner sub-assemblies are saved in the `Formwork tools` folder in the model folder.

If you want to use the conditions in other models, you can copy or move the files and the condition setup files to any of the system folders. To ensure that the conditions work in other models, all custom components that the conditions use must exist in the **Applications and components** catalog.

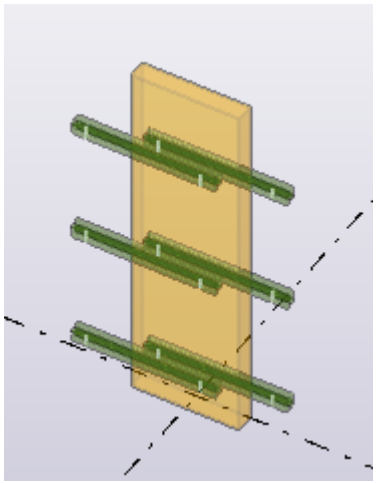
Create a new panel sub-assembly

The panel sub-assemblies are created using **Formwork panel sub-assembly wizard**.

The outcome of the wizard is stored in an external file with a name.

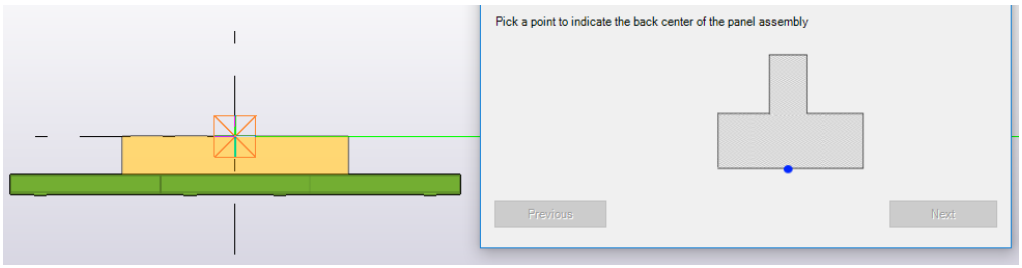
To open the dialog box, click in the **Condition setup** dialog box on a condition that requires a panel sub-assembly. Alternatively, you can search for the **Formwork panel sub-assembly wizard** in the **Applications and components** catalog.

Before you can start creating the panel sub-assemblies, you need to have a formwork panel and all necessary accessories in the model. When you have created the formwork structure, you can start the wizard.

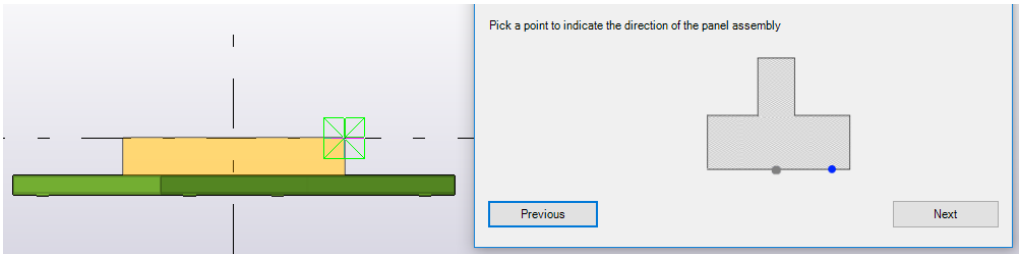


1. Pick the center point of the panel sub-assembly.

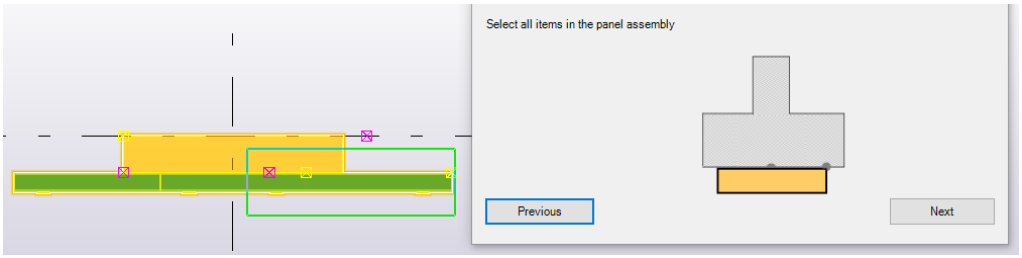
   This point will be placed exactly to the center of the concrete wall in the final condition placement. Note that after you have picked the point, the wizard steps forward automatically. You can step back to re-pick a new point.



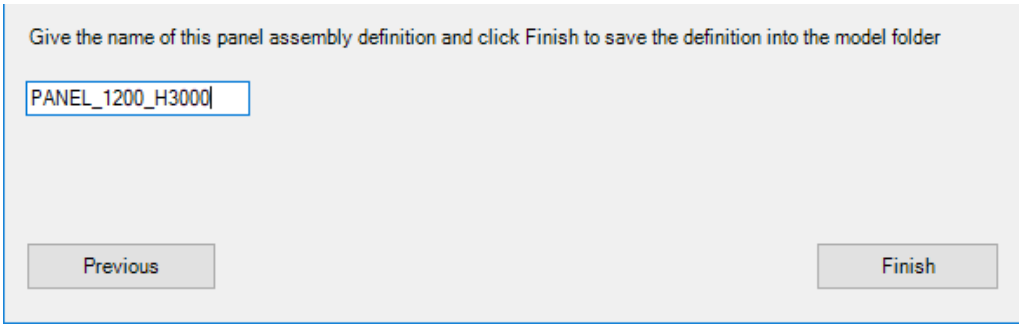
2. Pick a point showing the direction of the wall.



3. Select all formwork items belonging to the panel sub-assembly, and click **Next**.



4. Enter a file name to save the panel sub-assembly. Click **Finish** to close the wizard.

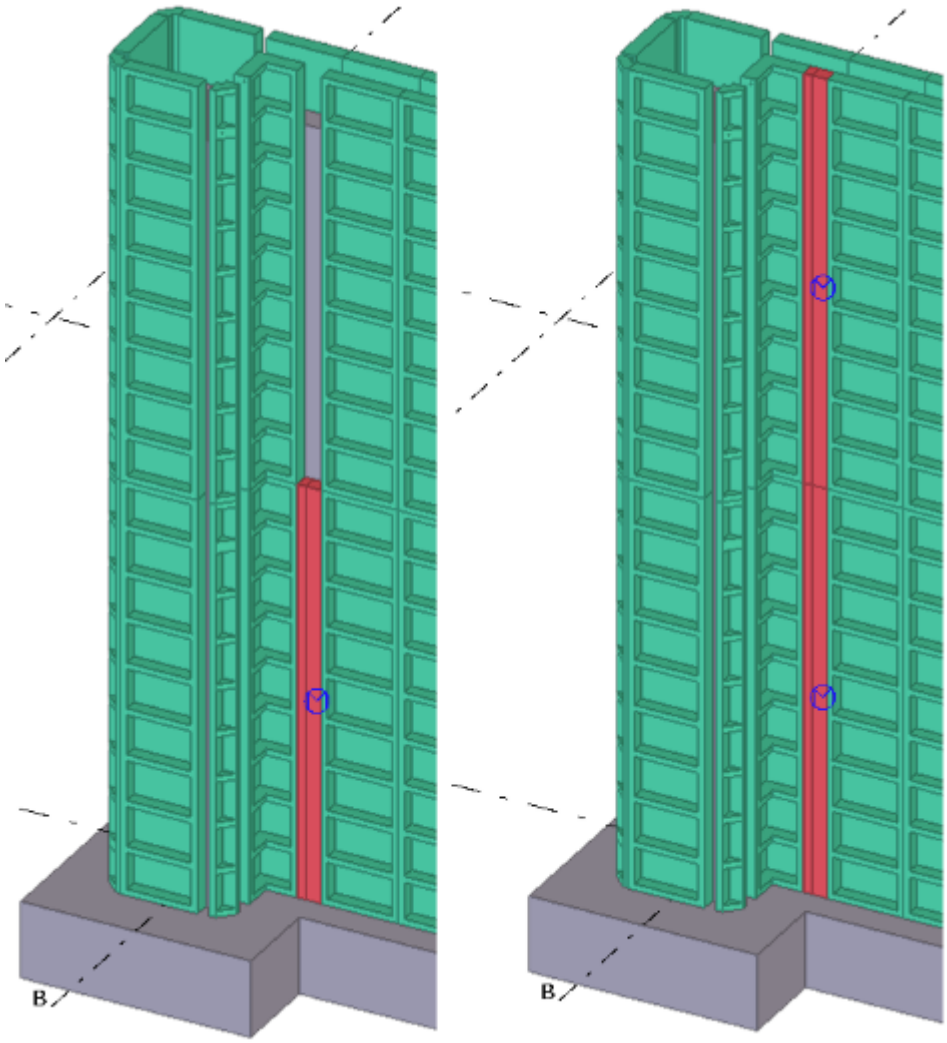


All panel sub-assemblies are saved in the `Formwork tools` folder in the model folder.

If you want to use the conditions in other models, you can copy or move the files and the condition setup files to any of the system folders. To ensure that the conditions work in other models, all custom components that the conditions use must exist in the **Applications and components catalog**.

Fillers tab

Use the **Fillers** tab to create filler items to fill the gaps between two formwork panels.



Note that this functionality does not work between condition and a panel, meaning that the gap at the end of the wall cannot be filled with this setup.

1. On the **Fillers** tab, select the filler in the **Filler configuration** list, and set the other filler properties.
2. Click **Apply** and either **Select single panel** or **Select two panels**, depending on how you want to insert the fillers.
   - To insert the filler by selecting a single panel, select the formwork panel and a point to indicate the location (left, right, top, bottom of the panel) and the width of the filler.
   - To insert the filler by selecting two adjacent panels, select the first formwork panel, and then select the second formwork panel.

Ties tab

Use the **Ties** tab to insert an array of ties between two formwork panels on the opposite sides of the wall.

1. On the **Ties** tab, select the tie in the **Tie type** list, and set the other tie properties.
2. To add ties manually, set **Tie layout** to **Single tie**, **Left side**, or **Right side**.
3. Click **Apply** and **Insert new** to start placing ties.
4. In the model, hover your mouse pointer over the panel and pick the points where you want to add ties.
5. Alternatively, to add ties automatically, set **Tie layout** to **Preset array**. Select the panel, click **Apply** and **Add to selected**. The ties are placed automatically.
6. If you want modify ties:
   a. To adjust tie position, select a tie array, and drag the handle to move the tie to the position you want.
   b. To add new ties, duplicate the existing ones. Hold down the **Ctrl** key, and drag a tie to a new position.
   c. To delete individual ties, select a tie and press the **Delete** key.
   d. To change the properties, spacing and/or start distances, modify the properties in the dialog box and click **Modify**.

Clamps tab

Use the **Clamps** tab to insert a linear array of clamps, locks, or brackets at the joining edges between two formwork panels to keep the formwork together. Clamp placing works in the same way as the tie placing.

1. On the **Clamps** tab, select the clamp in the **Clamp type** list, and set the other clamp properties.
2. To add clamps manually, set **Clamp layout** to **Single clamp**. With the **Custom array** option you can set the offset and spacing.
3. Click **Apply** and **Insert new** to start placing clamps.
4. In the model, hover your mouse pointer over the panel, and pick the points where you want to add clamps.
5. Alternatively, to add clamps automatically, set **Clamp layout** to **Preset array**. Select the panel, click **Apply** and **Add to selected**.

   The **Add accessories** dialog box opens where you can select which accessories are added with the clamps. Click **Add accessories to selected**. The clamps and other accessories are placed automatically to the selected panels.

6. If you want modify clamps:
    a. To adjust a clamp position, select a clamp, and drag the handle to move the clamp to the position you want.
    b. To add new clamps, duplicate the existing ones. Hold down the **Ctrl** key, and drag a clamp to a new position.
    c. To delete individual clamps, select a clamp and press the **Delete** key.
    d. To change the properties, spacing and/or start distances, modify the properties in the dialog box and click **Modify**.

Walers tab

Use the **Walers** tab to insert walers to formwork panels. Typically, a waler is used to connect two panels on top of each other but it can also be placed in one single panel. The waler typically consists of the waler beam and two lock devices fixing the waler to the panel.

1. On the **Walers** tab, select the waler in the **Waler type** list, and set the other waler properties.
2. Enter the longitudinal offset. Leave the value empty to get the waler beam centered between the input points.
3. Click **Apply** and **Insert new** to start placing walers.
4. In the model, hover your mouse pointer over the panel and pick the first preset point. This is the first input point of the waler beam and the location for the first lock device.
5. Move the mouse pointer on the next panel and pick the second preset point. This is the end point of the waler beam and the location for the second lock device.

    If you did not enter the longitudinal offset value, the waler beam is centered between the picked points. Otherwise, the waler beam starts at the given offset from the first picked point.

    Alternatively, to add walers automatically, click **Apply** and **Add to selected**. The **Add accessories** dialog box opens where you can select which accessories are added with the walers. Click **Add accessories to selected**. The walers and other accessories are placed automatically to the selected panels.
6. If you want modify walers:
    a. To adjust a waler position, select a waler, and drag the point handle to move the waler to the position you want.

        By holding down the **Alt** key, you can drag the point anywhere, even to a different panel.
    b. To change the longitudinal offset, drag the point handle between the picked points.

c. To add new walers, hold down the **Ctrl** key, and drag a main handle of a waler to a new position.

Braces tab

Use the **Braces** tab to insert supporting braces for a single formwork panel.

1. On the **Braces** tab, select the brace layout in the **Brace layout** list, and set the other brace properties.

   To add braces manually, set **Braces layout** to **Single brace**. With the **Custom array** option you can set the offset and spacing.
2. Click **Apply** and **Insert new** to start placing braces.
3. In the model, hover your mouse pointer over the panel and pick the points where you want to add braces.
4. Alternatively, to add braces automatically, set **Brace layout** to **Preset array**. Select the panel, click **Apply** and **Add to selected**. The **Add accessories** dialog box opens where you can select which accessories are added with the braces. Click **Add accessories to selected** and the braces and other accessories are placed automatically to the selected panels.

Platforms tab

Use the **Platforms** tab to insert an array of pour platforms at the top edge of formwork panels.

1. On the **Platforms** tab, select the platform type in the **Platform type** list, and set the platform spacing and offset.
2. Click **Apply** and **Insert new** to start placing platforms.
3. In the model, hover your mouse pointer over the panel and pick the start and end points to create a linear array of platforms along the top edge of the panel.
4. Alternatively, to add platforms automatically, select the panel and click **Add to selected**. Platforms are added automatically to the selected panels.

Formwork placing tools - Walls: configuration

You can set up your own configuration files for **Formwork placings tools - Walls**.

The configuration is done by using comma separated files (`.csv`), which can be edited with Microsoft Excel or any standard text editor. Each separate formwork sub-tool component has its own configuration file.

The configuration files can be located in any of the system folders, or in the sub-folder called `Formwork tools` in the current model folder.

Example configuration files are located in the `...\Trimble\Tekla Structures\<version>\Environments\common\system\CIP\Formwork` folder.

The configuration files are typically named by the formwork supplier and/or the product families. There can be any number of files, and the files are identified with a specific suffix. Each of the files controls one of the sub-tools in the component. The files contain varying number of columns.

Use the following configuration files for **Formwork placings tools - Walls** to set up

- panels: `xxxx.FormworkTools.Panels.csv`
- ties for the tie placing tool: `xxxx.FormworkTools.Ties.csv`
- tie spacers placed by the tie placing tool: `xxxx.FormworkTools.TieSpacers.csv`
- clamps: `xxxx.FormworkTools.Clamps.csv`
- braces: `xxxx.FormworkTools.Braces.csv`
- pouring platforms: `xxxx.FormworkTools.Platforms.csv`
- walers: `xxxx.FormworkTools.Walers.csv`
- conditions (corners, pilasters and bulkheads): `xxxx.FormworkTools.Conditions.csv`
- fillers: `xxxx.FormworkTools.Fillers.csv`

Accessories in formwork placing tools

You can set up any formwork accessory to be created as a beam, an item, or as a custom part. For beams and items you can additionally set up any of the beam or items properties or UDAs either in the configuration file by adding new columns, or as property files.

Each configuration file contains a header row and data rows. The header row is the first row that is not a comment row, and it gives the configuration parameter names (column name). Each data row gives one product and defines the parameters used when placing the beam, item or custom part.

In addition to header row and data rows the file can contain comment rows. A comment row is any row starting with text //.

You can define the distance unit by adding a row:

```
DISTANCE_UNIT=MM
```

When the distance unit is defined using the above setting, all distance values can be given as decimal values in the specified units. The supported units are: `MM`, `DN`, `CM`, `M`, `INCH`, `FEET`.

The following accessories are available for different **Formwork placings tools - Walls** product types. For example, for a brace you can specify `Brace`, `Bracket1` and `Bracket2` accessories.

Product type	Accessory
Panel	`Panel`
Tie	`Part1`, the actual tie rod `Part2`, the front lock `Part3`, the back lock

Tie spacer	`SpacerTube`
	`StartCone`
	`EndCone`

Clamp	`Part1`

Brace	`Brace`
	`Bracket1`
	`Bracket2`

Platform	`Part1` `Part2`

Waler	`Beam`
	`Clamp1`
	`Clamp2`

Filler	`Timber`
	`Stud`
	`Plywood`
	`Part1`
	`Part2 - Part10`

Common configuration parameter fields

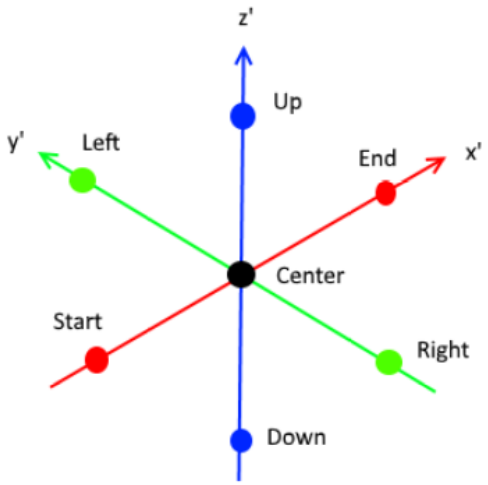
To configure the accessories in any of the formwork product types, define the following fields in the `.csv` configuration file.

The text `[Accessory]` in the following table denotes the accessory in question.

Column name in the .csv file	Description

`[Accessory].Profile`
`[Accessory].Item`
`[Accessory].CustomPart`
	Formwork placing tools can place either beams, items, or custom parts to the model.
	If all products are of the same model object type you can use one of the

Column name in the .csv file	Description
	three field names and give the corresponding data value. The field name specifies the model object default type but you can also override the model object type by adding the prefix `BEAM:`, `ITEM:` or `CUSTOMPART:` for the actual value. The actual value for `BEAM:` is a valid profile string, for `ITEM:` a valid item shape name, and for `CUSTOMPART:` a valid custom part name.
`[Accessory].Attributes`	Saved property file used to set either part, item or custom part properties.
`[Accessory].Name`	Name of the part or the item, or the **Name** property of the custom part.
`[Accessory].Class`	Class of the part or the item, or the **Class** property of the custom part.
`[Accessory].Material`	Material of the part or the item, or the **Material** property of the custom part.
`[Accessory].AssPrefix`	Part or item numbering assembly prefix, or the **AssPrefix** property of the custom part.
`[Accessory].AssStartNo`	Part or item assembly start number, or the **AssStartNo** property of the custom part.
`[Accessory].PartPrefix`	Part or item numbering part prefix, or the **PartPrefix** property of the custom part.
`[Accessory].PartStartNo`	Part or item part start number, or the **PartStartNo** property of the custom part.
`[Accessory].Finish`	Finish of the part or the item, or the **Finish** property of the custom part.
`[Accessory].InputOrder`	Set the final locations of the input points in relation to the default generic start and end points. The possible values are: • `StartEnd`: default if no other value is specified. • `EndStart`: reverses the default start/end points.

Column name in the .csv file	Description
	• `CenterStart`: the first input point is at the center of the start/end point and the second point is at the start point. • `CenterEnd`: the first input point is at the center of the start/end point and the second point is at the end point. • `CenterLeft`: the first input point is at the center of the start/end point and the second point is at the left point. • `CenterRight`: the first input point is at the center of the start/end point and the second point is at the right point. • `CenterUp`: the first input point is at the center of start/end point and the second point is up from the center point. • `CenterDown`: the first input point is at the center of the start/end point and the second point is down from the center point. Note that if the `.csv` file does not contain any value, the default value `StartEnd` is used. 

Column name in the .csv file	Description
[Accessory].PlanePosition	This is **Position in plane** when the beam, item, or custom part is inserted in the plane view. The options are MIDDLE, LEFT, and RIGHT.
[Accessory].PlaneOffset	Offset on plane. The default value is 0.
[Accessory].Rotation	This is **Rotation** when the custom part is inserted in the plane view. The options are FRONT, TOP, BACK, and BELOW.
[Accessory].RotationOffset	Rotation offset in degrees. The default value is 0.
[Accessory].DepthPosition	This is **Position at depth** when the beam, item, or custom part is inserted in the plane view. The options are MIDDLE, FRONT, and BEHIND.
[Accessory].DepthOffset	Offset at depth. The default value is 0.
[Accessory].StartOffset	Offset of the actual start point from the generic start point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
[Accessory].EndOffset	Offset of the actual end point from the generic end point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
[Accessory].UDA.XXXXXX	Define additional UDA values applied to the model objects. You can introduce as many UDA values as needed. Note that the UDA name (XXXXX) must be the internal name, not the

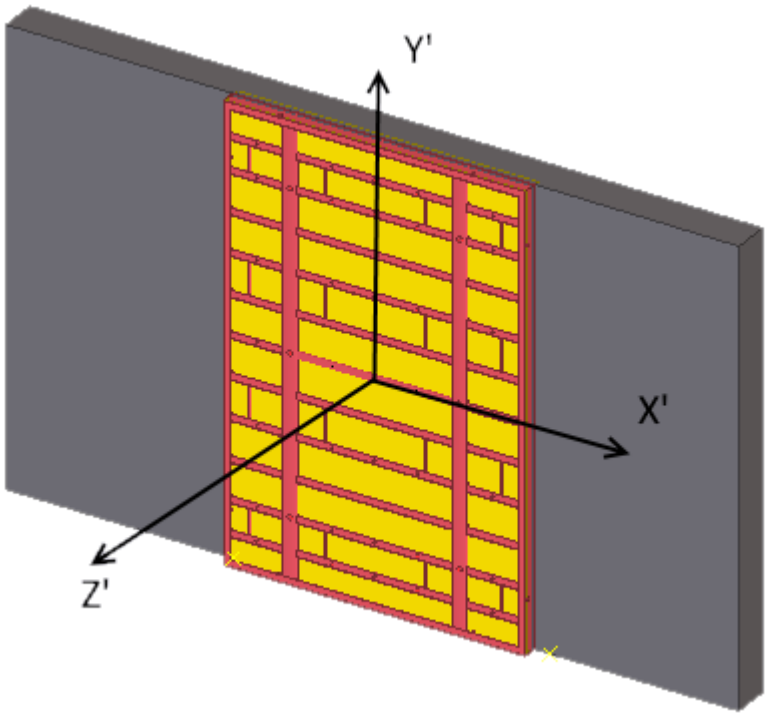
Column name in the .csv file	Description
	localized name shown in the user interface.

Configure the panel placing tool

`.FormworkTools.Panels.csv`

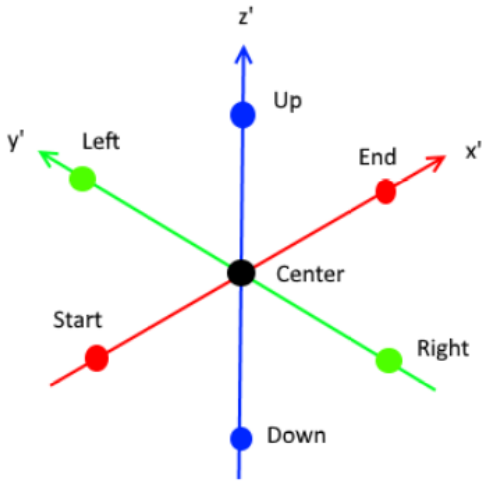
Add or modify one row for each panel.

The configuration of all accessories that can be added to panels is done in the local coordinate system of the panel. The origin of the panel is located at the center point of the panel.



Definition	Description
`Supplier`	Supplier name, this is typically the same for all rows. The name is shown in the **Formwork supplier** list.
`Family`	Family name, which is shown in the **Product family** list.
`Name`	Unique panel name, which is shown in the **Formwork panel** list.
`PanelName`	Name of the custom part that will be inserted in the model.

Definition	Description
`PanelAttributes`	Property file saved in the component dialog box to be used when the custom part will be inserted in the model.
`PanelInputOrder`	Define the orientation of the custom part panels by specifying where the start and end handles of the custom part will go. Set the final locations of the input points in relation to the default generic start and end points. The possible values are: • `StartEnd`: default if no other value is specified. • `EndStart`: reverses the default start/end points. • `CenterStart`: the first input point is at the center of the start/end point and the second point is at the start point. • `CenterEnd`: the first input point is at the center of the start/end point and the second point is at the end point. • `CenterLeft`: the first input point is at the center of the start/end point and the second point is at the left point. • `CenterRight`: the first input point is at the center of the start/end point and the second point is at the right point. • `CenterUp`: the first input point is at the center of start/end point and the second point is up from the center point. • `CenterDown`: the first input point is at the center of the start/end point and the second point is down from the center point.

Definition	Description
	Note that if the `.csv` file does not contain any value, the default value `StartEnd` is used. 
`PanelPlanePosition`	Use the following position values to define the custom part's position, offset and rotation in relation to the custom part start and end points. Use **Position in plane** to insert the custom part in plane view. The options are **MIDDLE**, **LEFT**, and **RIGHT**.
`PanelPlaneOffset`	Offset on plane. The default value is 0.
`PanelRotation`	Use **Rotation** to insert the custom part in the plane view. The options are **FRONT**, **TOP**, **BACK**, and **BELOW**.
`PanelRotationOffset`	Rotation offset in degrees. The default value is 0.
`PanelDepthPosition`	Use **Position at depth** to insert the custom part in plane view. The options are **MIDDLE**, **FRONT**, and **BEHIND**.
`PanelDepthOffset`	Offset at depth. The default value is 0.
`PanelStartOffset`	`PanelStartOffset` and `PanelEndOffset` change the

Definition	Description
	location of the actual start and end handles of the custom part. `PanelStartOffset` is the offset of the actual start point from the generic start point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
`PanelEndOffset`	`PanelEndOffset` is the offset of the actual end point from the generic end point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
`HeightProperty`	Name of the height property in the custom part. If the height is fixed, this is empty.
`HeightValue`	Height of the panel. Note that the height value is read from the `.csv` file.
`WidthProperty`	Name of the width property in the custom part. If the width is fixed, this is empty.
`WidthValue`	Width of the panel. Note that the width value is read from the `.csv` file.
`ThicknessProperty`	Name of the thickness property in the custom part. If the thickness is fixed, this is empty.
`ThicknessValue`	Thickness of the panel. Note that the thickness value is read from the `.csv` file.
`TieX`	`TieX` specifies the x locations of the ties. Values must be enclosed in parentheses () and separated by spaces.
`TieY`	`TieY` specifies the y locations of the ties. Values must be enclosed in

Definition	Description
	parentheses () and separated by spaces.

`TiePickX`
`TiePickY`
or
`TiePickXY`
	Define the possible x and y locations of the ties when placing and picking a single tie. You can either specify the x and y values separately in two value fields to define a regular grid of locations, or if your panels require an irregular pattern, you can specify a list of x and y value pairs.
	Always define the x and y values in two separate fields (`TiePickX` and `TiePickY`), or x and y value pairs in one field (`TiePickXY`).
	Values must be enclosed in parentheses () and separated by spaces.

`ClampX`	`ClampX` specifies the x locations of the clamps at the horizontal top/ bottom edges. Values must be enclosed in parentheses () and separated by spaces.
`ClampY`	`ClampY` specifies the y locations of the clamps at the vertical left/right edges. Values must be enclosed in parentheses () and separated by spaces.

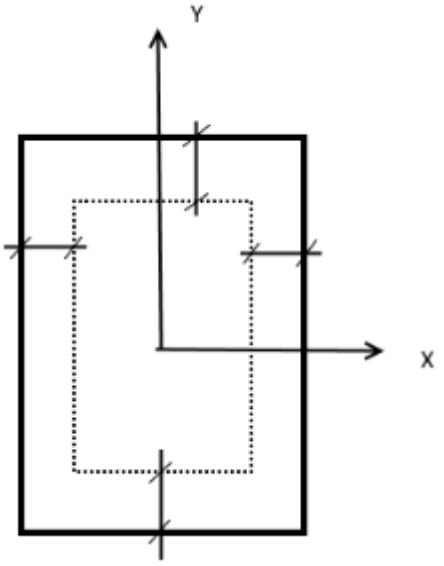
`ClampPickX`
`ClampPickY`
or
`ClampPickXY`
	Define the possible x and y locations of the clamps when placing and picking a single clamp. You can either specify the x and y values separately in two value fields to define a regular grid of locations, or if your panels require an irregular pattern, you can specify a list of x and y value pairs.
	Always define the x and y values in two separate fields (`ClampPickX` and `ClampPickY`), or x and y value pairs in one field (`ClampPickXY`).
	Values must be enclosed in parentheses () and separated by spaces.

Definition	Description

`BraceX`
`BraceY`
or
`BraceXY`
	Define the preset pattern for the braces. You can either specify the x and y values separately in two value fields to define a regular grid of locations, or if your panels require an irregular pattern, you can specify a list of x and y value pairs.
	Always define the x and y values in two separate fields (`BraceX` and `BraceY`), or x and y value pairs in one field (`BraceXY`).
	Values must be enclosed in parentheses () and separated by spaces.

`BraceTiltedX`
`BraceTiltedY`
or
`BraceTiltedXY`
	Define the preset pattern for the braces when the panel is tilted. You can either specify the x and y values separately in two value fields to define a regular grid of locations, or if your panels require an irregular pattern, you can specify a list of x and y value pairs.
	Always define the x and y values in two separate fields (`BraceTiltedX` and `BraceTiltedY`), or x and y value pairs in one field (`BraceTiltedXY`).
	Values must be enclosed in parentheses () and separated by spaces.

`BracePickX`
`BracePickY`
or
`BracePickXY`
	Define the possible x and y locations of the braces when placing and picking a single brace in a panel. You can either specify the x and y values separately in two value fields to define a regular grid of locations, or if your panels require an irregular pattern, you can specify a list of x and y value pairs.
	Always define the x and y values in two separate fields (`BracePickX` and

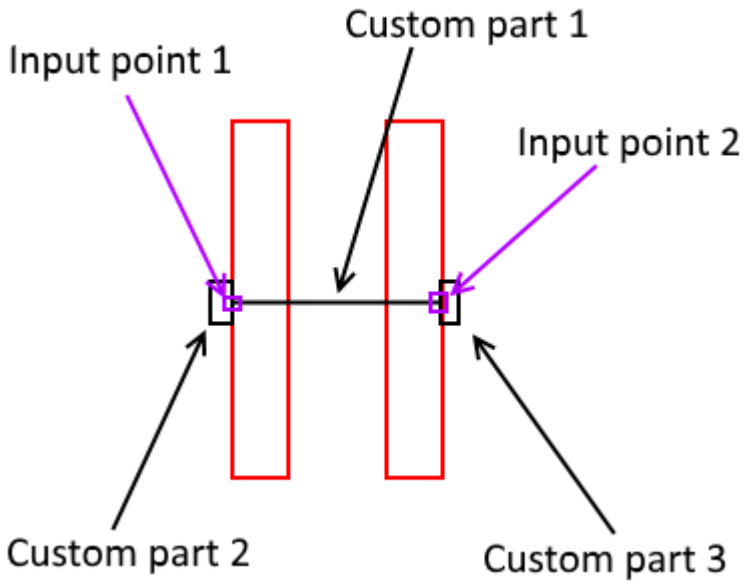
Definition	Description
	BracePickY), or x and y value pairs in one field (BracePickXY). Values must be enclosed in parentheses () and separated by spaces.
BraceTiltedPickX BraceTiltedPickY or BraceTiltedPickXY	Define the possible x and y locations of the braces when placing and picking a single brace in a tilted panel. You can either specify the x and y values separately in two value fields to define a regular grid of locations, or if your panels require an irregular pattern, you can specify a list of x and y value pairs. Always define the x and y values in two separate fields (BraceTiltedPickX and BraceTiltedPickY), or x and y value pairs in one field (BraceTiltedPickXY). Values must be enclosed in parentheses () and separated by spaces.
WalerEdgeOffset	Offset of the waler input point (=location of the clamp 1 and/or clamp 2) from the panel edge when the walers are placed automatically. 

Definition	Description
`WalerX`	X locations used for placing walers automatically at horizontal top/ bottom edges. The vertical (y) location is defined by `WalerEdgeOffset` from the top/bottom edge. Values must be enclosed in parentheses () and separated by commas or spaces.
`WalerY`	Y locations used for placing walers automatically at vertical left/right edges. The horizontal (x) location is defined by `WalerEdgeOffset` from left/right edge. Values must be enclosed in parentheses () and separated by commas or spaces.
`WalerPickX`	Define the possible x locations of the walers when placing and picking a single waler. Values must be enclosed in parentheses () and separated by commas or spaces.
`WalerPickY`	Define the possible y locations of the walers when placing and picking a single waler. Values must be enclosed in parentheses () and separated by commas or spaces.

Configure the tie placing tool

`xxxx.FormworkTools.Ties.csv`

A generic tie consists of three custom components: a bolt that usually goes through the wall, and two locks on both sides of the formwork.



Add or modify one row for each tie.

Definition	Description
`Supplier`	Supplier name, which is typically the same for all rows. The name is shown in the **Formwork supplier** list.
`Family`	Family name, which is shown in the **Product family** list.
`Name`	Tie name, which is shown in the **Tie type** list. This name can be unique, or multiple tie configurations may have the same name. When multiple tie configurations have the same name, the tool automatically selects the first suitable tie based on the wall thickness.
`MaxLength`	Maximum length of the tie, measured from outer faces of the two formwork panels on opposite sides.
`LengthProperty`	If the tie custom part is parametric and has a parameter controlling the effective length, this is the name of the **Length** property.

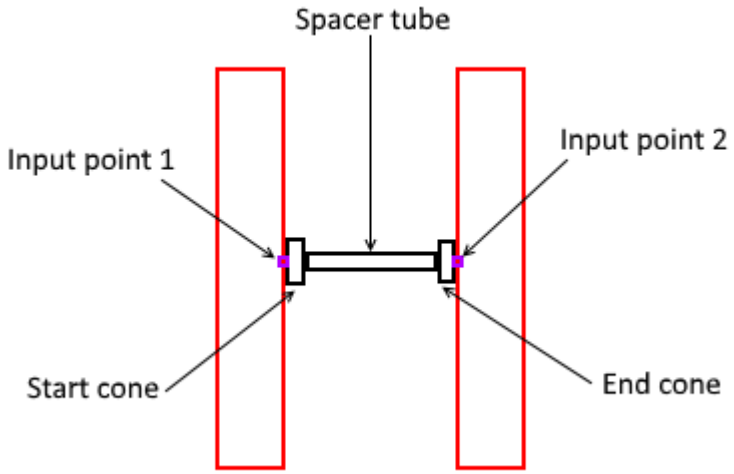
Definition	Description
`Part1Name`	Name of the custom part 1 that will be inserted in the model (=`bolt`).
`Part1Attributes`	Property file saved in the component dialog box to be used when the custom part is inserted in the model.
`Part1PlanePosition`	This is **Position in plane** when the custom part is inserted in the face view of the main formwork panel.
`Part1PlaneOffset`	This is **Offset on plane** value when the custom part is inserted in the face view of the main formwork panel.
`Part1Rotation`	This is **Rotation** when the custom part is inserted in the face view of the main formwork panel.
`Part1RotationOffset`	This is **Rotation offset** when the custom part is inserted in the face view of the main formwork panel.
`Part1DepthPosition`	This is **Position at depth** when the custom part is inserted in the face view of the main formwork panel.
`Part1DepthOffset`	This is **Offset at depth** when the custom part is inserted in the face view of the main formwork panel.
`Part1StartOffset`	Offset of the first insertion point in relation to **Input point 1**.
`Part1EndOffset`	Offset of the first insertion point in relation to **Input point 2**.
`Part2Name`	Name of the custom part 2, which typically is the lock on the front side of the main formwork panel.
`Part2Attributes`	Property file saved in the component dialog box to be used when the custom part will be inserted in the model.
`Part2StartOffset`	Offset of the second insertion point in relation to **Input point 1**.
`Part2EndOffset`	Location of the second insertion point in relation to the **Input point 1**.
`Part2...`	For custom part 2, you can specify the same fields as for custom part 1.
`Part3Name`	Name of the custom part 3, which typically is the lock on the back side of the main formwork panel.

Definition	Description
`Part3Attributes`	Property file saved in the component dialog box to be used when the custom part will be inserted in the model.
`Part3StartOffset`	Offset of the first insertion point in relation to **Input point 2**.
`Part3EndOffset`	Location of the second insertion point in relation to the **Input point 2**.
`Part3...`	For custom part 3, you can specify the same fields as for custom part 1.

Configure the spacers for the tie placing tool

`xxxx.FormworkTools.TieSpacers.csv`

A generic tie spacer consists of three custom components: a tube that usually goes through the wall, and two optional cones at each side of the wall.



Add or modify one row for each tie spacer.

Definition	Description
`Supplier`	Supplier name, which is typically the same for all rows. The name is shown in the **Formwork supplier** list.
`Family`	Family name, which is shown in the **Product family** list.

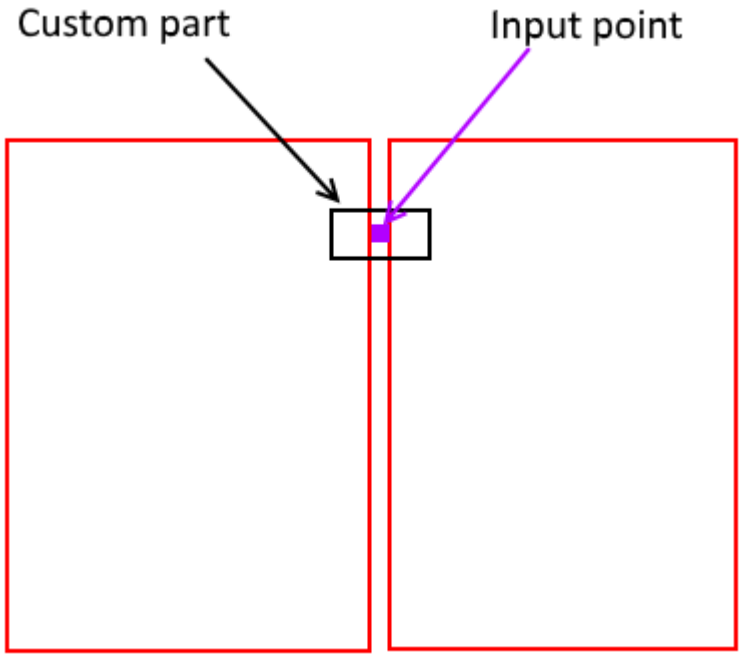
Definition	Description
`Name`	Spacer name, which is shown in the **Spacer name** list. This name can be unique, or multiple tie spacer configurations may have the same name. When multiple tie spacer configurations have the same name, the tool automatically selects the first suitable tie spacer based on the wall thickness.
`MinLength`	Minimum possible length of the tube, measured from the inner faces of the two formwork panels on the opposite sides.
`MaxLength`	Maximum possible length of the tube, measured from the inner faces of the two formwork panels on the opposite sides.
`LengthProperty`	If the tie spacer custom part is parametric and has a parameter controlling the effective length, this is the name of the **Length** property.
`SpacerTubeName`	Name of the custom part (=`tube`).
`SpacerTubeAttributes`	Property file saved in the component dialog box to be used when the custom part will be inserted in the model.
`SpacerTubePlanePosition`	This is **Position in plane** when the custom part is inserted in the face view of the main formwork panel.
`SpacerTubePlaneOffset`	This is **Offset in plane** when the custom part is inserted in the face view of the main formwork panel.
`SpacerTubeRotation`	This is **Rotation** when the custom part is inserted in the face view of the main formwork panel.
`SpacerTubeRotationOffset`	This is **Rotation offset** when the custom part is inserted in the face view of the main formwork panel.
`SpacerTubeDepthPosition`	This is **Position in depth** when the custom part is inserted in the face view of the main formwork panel.

Definition	Description
SpacerTubeDepthOffset	This is **Offset in depth** when the custom part is inserted in the face view of the main formwork panel.
SpacerTubeStartOffset	Offset of the first insertion point in relation to **Input point 1**.
SpacerTubeEndOffset	Offset of the first insertion point in relation to **Input point 2**.
StartConeName	Name of the custom part, which typically is the cone on the front side of the main formwork panel.
StartConeAttributes	Property file saved in the component dialog box to be used when the custom part is inserted in the modell.
StartConeStartOffset	Offset of the first insertion point in relation to **Input point 1**.
StartConeEndOffset	Location of the second insertion point in relation to the **Input point 1**.
StartCone...	For custom part 2, you can specify the same fields as for custom part 1.
EndConeName	Name of the custom part which typically is the cone on the back side of the main formwork panel.
EndConeAttributes	Property file saved in the component dialog box to be used when the custom is be inserted in the model.
EndConeStartOffset	Offset of the first insertion point in relation to **Input point 2**.
EndConeEndOffset	Location of the second insertion point in relation to the **Input point 2**.
EndCone...	For custom part 3, you can specify the same fields as for custom part 1.

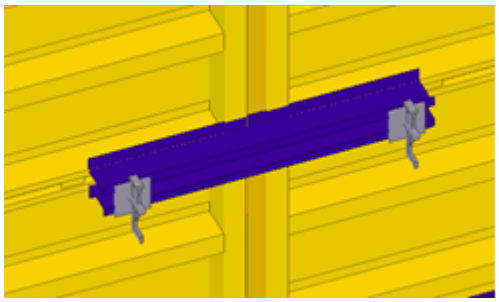
Configure the clamp placing tool

xxxx.FormworkTools.Clamps.csv

The generic clamp contains one custom part. The input points are at the edges of the two panels and thus the clamp can be parametric in terms of the filler space, if needed.



Note that to create the brackets with some additional bolts or locks, you need to make a new custom part containing both the bracket and all necessary fittings:



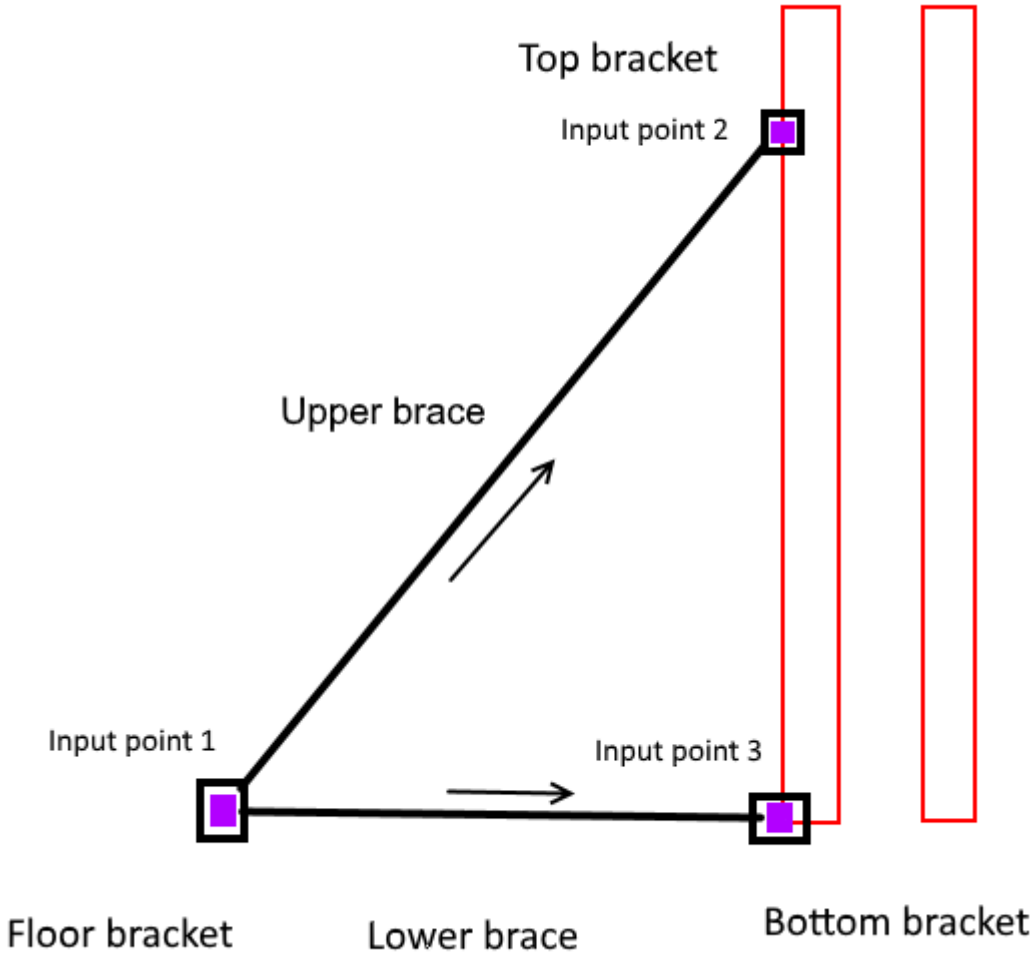
Definition	Description
`Supplier`	Supplier name, which is typically the same for all rows. The name is shown in the **Formwork supplier** list.
`Family`	Family name, which is shown in the **Product family** list.
`Name`	Unique clamp name.
`CenterInput`	When `YES`, the first insertion point is in the middle of the seam of the two panels. If the value is `NO`, the insertion

Definition	Description
	points are offset from the edge of the panels.
FillerGapProperty	If the clamp custom part is parametric, and the parameter can adjust the gap between two panels, this is the name of that parameter. This field is empty if the clamp is not parametric.
MaxFillerGap	This value is used when clamps or walers are added automatically along with a formwork filler. If the free space (=width of the filler) is less or equal to the given MaxFillerGap, clamps will be inserted. If the free space is greater than the given MaxFillerGap, walers will be inserted.
Part1Name	Name of the clamp custom part 1.
Part1Attributes	Property file saved in the component dialog box to be used when the custom part is inserted in the model.
Part1PlanePosition	This is **Position in plane** when the custom part is inserted in the face view of the main formwork panel.
Part1PlaneOffset	This is **Offset in plane** when the custom part is inserted in the face view of the main formwork panel.
Part1Rotation	This is **Rotation** when the custom part is inserted in the face view of the main formwork panel.
Part1RotationOffset	This is **Rotation offset** when the custom part is inserted in the face view of the main formwork panel.
Part1DepthPosition	This is **Position in depth** when the custom part is inserted in the face view of the main formwork panel.
Part1DepthOffset	This is **Offset in depth** when the custom part is inserted in the face view of the main formwork panel.
Part1StartOffset	Offset of the first insertion point in relation to **Input point 1**.
Part1EndOffset	Location of the second insertion point in relation to the **Input point 1**.

Configure the brace placing tool

`xxxx.FormworkTools.Braces.csv`

A generic brace may consists of five custom parts, which are inserted in the model by using three input points. You can select the upper and lower braces separately. In the brace configuration file, the upper and lower braces are defined separately; they both have own rows.



Definition	Description
`Supplier`	Supplier name, which is typically the same for all rows. The name is shown in the **Formwork supplier** list.
`Family`	Family name, which is shown in the **Product family** list.

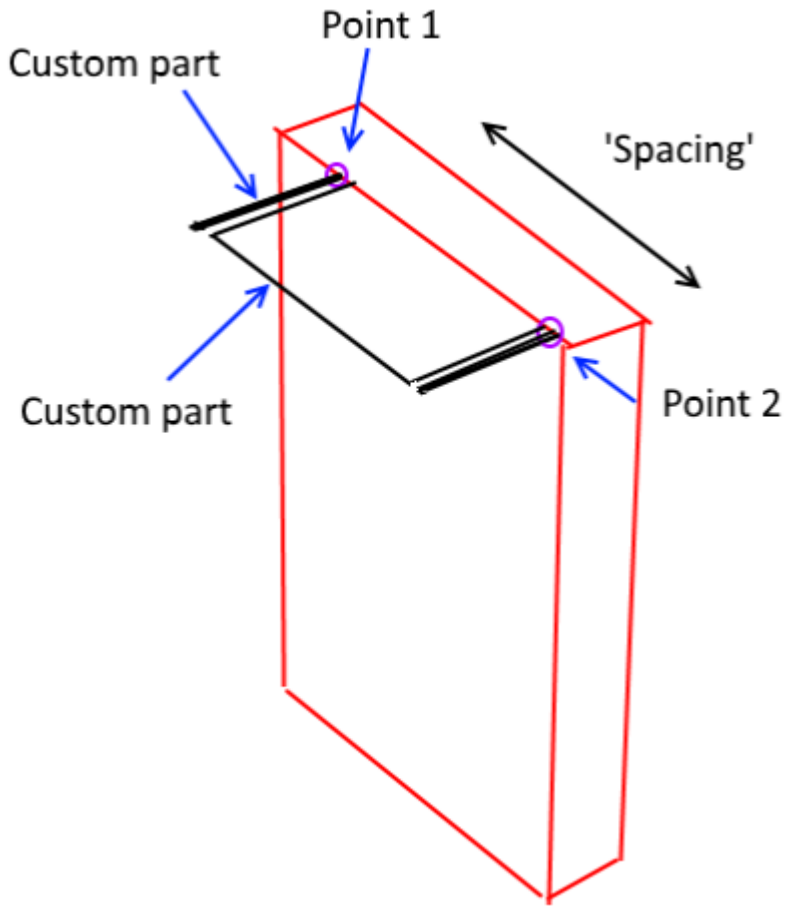
Definition	Description
`Name`	Brace name which is shown in the list. If you have several rows with the same name, the tool automatically selects the suitable brace based on the minimum and maximum length.
`Type`	Type of the brace. The options are `Upper` or `Lower`.
`MinLength`	Minimum possible length of the main brace measured from **Input point 1** to **Input point 2**.
`MaxLength`	Maximum possible length of the main brace measured from **Input point 1** to **Input point 2**.
`LengthProperty`	If the brace custom part is parametric and has a parameter controlling the effective length, this is the name of the **Length** property. At insertion it will get the actual length value between **Input point 1** and **Input point 2**.
`BraceName`	Name of the custom part for the upper or lower brace which will be inserted into the model.
`BraceAttributes`	Property file saved in the component dialog box to be used when the custom part is inserted in the model.
`BracePlanePosition`	This is **Position in plane** when the custom part is inserted in plane view.
`BracePlaneOffset`	This is **Offset in plane** when the custom part is inserted in plane view.
`BraceRotation`	This is **Rotation** when the custom part is inserted in plane view.
`BraceRotationOffset`	This is **Rotation offset** when the custom part is inserted in plane view.
`BraceDepthPosition`	This is **Position in depth** when the custom part is inserted in plane view.
`BraceDepthOffset`	This is **Offset in depth** when the custom part is inserted in plane view.
`BraceStartOffset`	Offset of the first insertion point in relation to **Input point 1**.
`BraceEndOffset`	Offset of the first insertion point in relation to **Input point 2**.

Definition	Description
`Bracket1Name`	Name of the custom part for the top bracket (upper brace) or bottom bracket (lower brace) which will be inserted into the model.
`Bracket1Attributes`	Property file saved in the component dialog box to be used when the custom part is inserted in the model.
`Bracket1StartOffset`	Offset of the first insertion point in relation to **Input point 1**.
`Bracket1EndOffset`	Location of the second insertion point in relation to the **Input point 3**.
`Bracket1...`	For custom part 2, you can specify the same fields as for custom part 1.
`Bracket2Name`	Name of the custom part for the floor bracket which will be inserted into the model. If both upper and lower brace have a definition for the bracket, the bracket defined for the lower brace will be created.
`Bracket2Attributes`	Property file saved in the component dialog box to be used when the custom part will be inserted in the model.
`Bracket2StartOffset`	Offset of the first insertion point in relation to **Input point 2**.
`Bracket2EndOffset`	Location of the second insertion point in relation to the **Input point 2**.
`Bracket2...`	For custom part 3, you can specify the same fields as for custom part 1.

Configure the pouring platform placing tool

`xxxx.FormworkTools.Platforms.csv`

A generic pouring platform contains two custom parts, which are inserted in the model as a linear array of custom parts.



Definition	Description
`Supplier`	Supplier name, which is typically the same for all rows. The name is shown in the **Formwork supplier** list.
`Family`	Family name, which is shown in the **Product family** list.
`Name`	Unique pouring platform name.
`Type`	The type of the platform (A or B). The length of the array is defined with start and end points of the platform. The custom part placing has two methods: • Type A: the length of one platform custom part is fixed, and the custom parts are inserted sequentially.

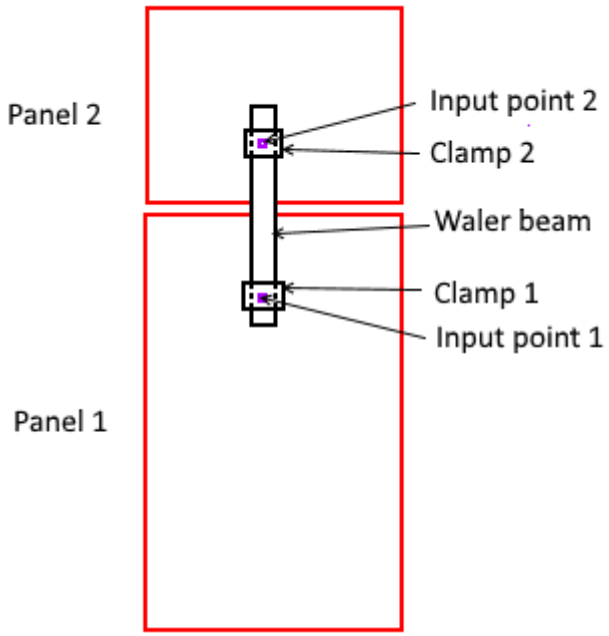
Definition	Description
	• Type B: the first custom part is a support, which is placed at start, intermediate and end points of the array. The second custom part is placed between the main support parts.
`Length`	Effective length of one platform element.
`Width`	Width of the platform.
`Spacing`	Spacing between the custom parts in the array.
`Part1Name`	Name of the custom part 1.
`Part1Attributes`	Property file saved in the component dialog box to be used when the custom part is inserted in the model.
`Part1PlanePosition`	This is **Position in plane** when the custom part is inserted in the face view of the main formwork panel.
`Part1PlaneOffset`	This is **Offset in plane** when the custom part is inserted in the face view of the main formwork panel.
`Part1Rotation`	This is **Rotation** when the custom part is inserted in the face view of the main formwork panel.
`Part1RotationOffset`	This is **Rotation offset** when the custom part is inserted in the face view of the main formwork panel.
`Part1DepthPosition`	This is **Position in depth** when the custom part is inserted in the face view of the main formwork panel.
`Part1DepthOffset`	This is **Offset in depth** when the custom part is inserted in the face view of the main formwork panel.
`Part1StartOffset`	Offset of the first insertion point in relation to **Input point 1**.
`Part1EndOffset`	Offset of the first insertion point in relation to **Input point 1**.
`Part2Name`	Name of the custom part 2.
`Part2Attributes`	Property file saved in the component dialog box to be used when the custom part is inserted in the model.

Definition	Description
`Part2StartOffset`	Offset of the first insertion point in relation to **Input point 1**.
`Part2EndOffset`	Location of the second insertion point in relation to the **Input point 1**.
`Part2...`	For custom part 2, you can specify the same fields as for custom part 1.

Configure the waler placing tool

`xxxx.FormworkTools.Walers.csv`

A generic waler consists of three custom parts, the waler beam and two clamp devices. The generic input points are located in the outer face of the panel.



Definition	Description
`Supplier`	Supplier name, which is typically the same for all rows. The name is shown in the **Formwork supplier** list.
`Family`	Family name, which is shown in the **Product family** list.
`Name`	Waler name, which is shown in the **Waler type** list. This name can be unique, or multiple tie waler configurations may have the same name. When multiple waler

Definition	Description
	configurations have the same name, the tool automatically selects the shortest waler based on the actual length measured between the input points. Typically, you can specify the same waler with a unique name and a common name. This way you can select the common name and let the system select the waler based on the input length. By selecting the unique name you can force the usage of a certain waler configuration.
`MaxLength`	Maximum possible effective length of the waler, measured from the first input point to the second input points (=clamp locations), typically little less than the physical length.
`MinLength`	Minimum length of a waler with variable length. For fixed length walers, leave the value empty or give same value as for the maximum length.
`BeamName`	Name of the clamp custom part 1.
`BeamAttributes`	Property file saved in the component dialog box to be used when the custom part is inserted in the model.
`BeamPlanePosition`	This is **Position in plane** when the custom part is inserted in the face view of the main formwork panel.
`BeamPlaneOffset`	This is **Offset in plane** when the custom part is inserted in the face view of the main formwork panel.
`BeamRotation`	This is **Rotation** when the custom part is inserted in the face view of the main formwork panel.
`BeamRotationOffset`	This is **Rotation offset** when the custom part is inserted in the face view of the main formwork panel.
`BeamDepthPosition`	This is **Position in depth** when the custom part is inserted in the face view of the main formwork panel.

Definition	Description
`BeamDepthOffset`	This is **Offset in depth** when the custom part is inserted in the face view of the main formwork panel.
`BeamStartOffset`	Offset of the first insertion point in relation to **Input point 1**.
`BeamEndOffset`	Location of the second insertion point in relation to the **Input point 1**.
`Clamp1Name`	Name of the clamp 1 custom part.
`Clamp1Attributes`	Property file saved in the component dialog box to be used when the custom part will be inserted in the model.
`Clamp1StartOffset`	Offset of the first insertion point in relation to **Input point 1**.
`Clamp1EndOffset`	Location of the second insertion point in relation to the **Input point 1**.
`Clamp1...`	For custom clamp 1, you can specify the same fields as for waler beam custom part.
`Clamp2Name`	Name of the clamp 2 custom part.
`Clamp2Attributes`	Property file saved in the component dialog box to be used when the custom part is inserted in the model.
`Clamp2StartOffset`	Offset of the first insertion point in relation to **Input point 1**.
`Clamp2EndOffset`	Location of the second insertion point in relation to the **Input point 1**.
`Clamp2...`	For custom clamp 2, you can specify the same fields as for waler beam and clamp 1 custom parts.

Configure the conditions for condition placing tool

`xxxx.FormworkTools.Conditions.csv`

A condition in the context of the formwork tools covers the L, T and X corners, pilasters, pour stops and columns. The final condition formwork is divided into two or more sub-assemblies depending on the condition type.

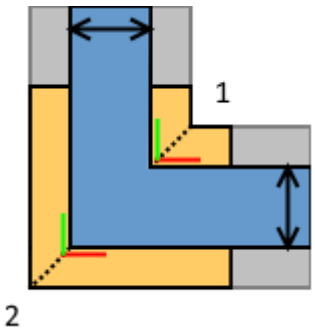
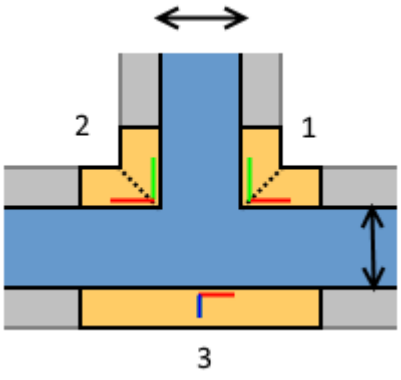
If you define condition configurations manually:

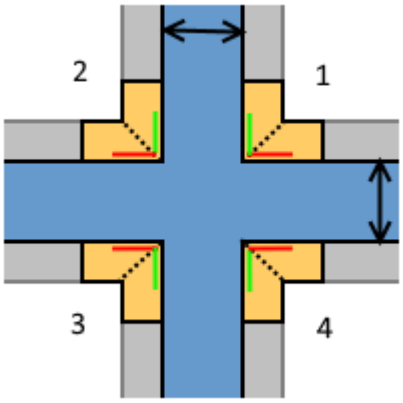
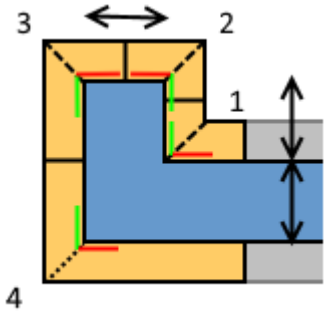
1. Model the full condition assembly using the custom parts, beams and/or filler tools.
2. Create the necessary sub-assemblies using the two wizards. Split the content of the complete condition to named sub-assemblies. When these

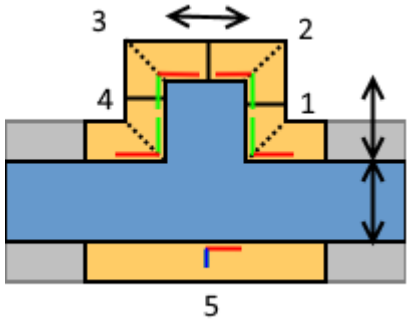
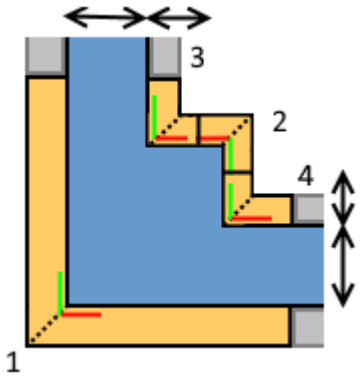
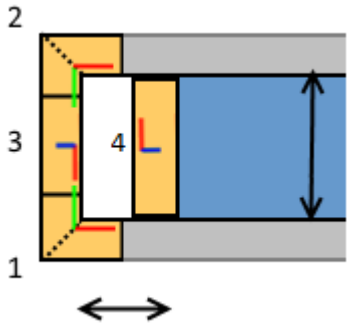
sub-assemblies are inserted in the model, the final location is parametric in the sense that the location depends on the actual geometry of the concrete structure.

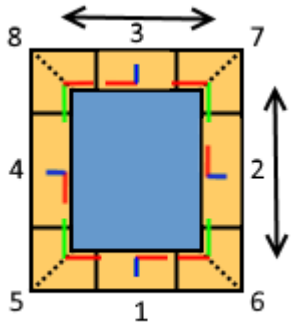
3. Insert a new condition specification in the condition configuration file.

The formwork condition placing tool supports the following condition types:

Definition	Description
`L corner (L)`	For L corner, you can specify the inner corner sub-assembly (1) and the exterior corner sub-assembly (2).  The corner sub-assembly is defined with the **Corner sub-assembly wizard**. With the wizard you can specify two sets of formwork items, which will be rotated according to the joining walls when the corner is placed in the model. The two corner sub-assemblies 1 and 2 are inserted according to the thickness of the joining walls.
`T corner (T)`	For T corner, you can specify two inner corners (1+2) and the back panel (3). 

Definition	Description
	The corner sub-assembly is defined with the **Formwork corner sub-assembly wizard**. The back panel sub-assembly is defined with the **Formwork sub-assembly wizard**, and it contains a single insertion point + direction. When the T corner is placed in the model, the insertion point of the panel sub-assembly is located at the intersection of the joining wall center line and the exterior face of the horizontal wall.
`X corner (X)`	For X corner, you can specify the four inner corners (1-4). 
`Corner pilaster (CP)`	For corner pilaster, you can specify one inner corner sub-assembly (1) and three external corner sub-assemblies (2-4). 
`Pilaster (P)`	For pilaster, you can specify two inner corners (1 and 4), two external

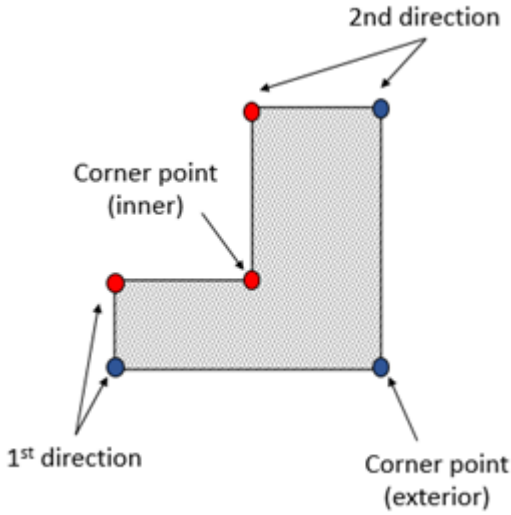
Definition	Description
	corners (2 and 3), and the back panel (5). 
`Pilaster inside corner (IP)`	For inner pilaster, you can specify one external corner (1) and three inner corners (2-3). 
`Bulkhead (B)`	For bulkhead, you can specify two external corners (1 and 2) and two middle panels (3 and 4). Panel 4 will be located at end of pour and all other sub-items will be offset by the given dimension. 

Definition	Description
Column (COL)	For column, you can specify four panel assemblies and/or four corner assemblies. 

Create a corner sub-assembly

Corner sub-assembly is the building block for one side of the L corner or any of the inner corners in the T and X corners.

1. First model all the needed formwork objects. Only beams and/or custom parts can be used for modeling, so if you have other components, you need to explode them.
2. Decide which object should rotate with the first direction, and which objects should rotate with the second edge at the corner if/when the corner angle varies.
3. Start the **Formwork corner sub-assembly wizard** tool.
4. Pick the corner point as shown below.



5. Pick a point to identify the first direction.

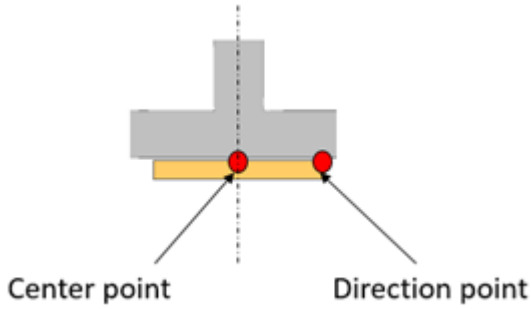
6. Select the corner parts (custom parts and/or beams) at the first side of the corner.
7. Pick a point to identify the second direction.
8. Select the corner parts (custom parts and/or beams) at the second side of the corner.
9. Enter a name and click **Finish** to save the detailed half of the corner into an external file.

Note that the wizard should be run separately for the inner corner (red dots) and external corner (blue dots). Ensure that you do not include same objects twice as that would lead to duplicate objects in the model.

Create a panel sub-assembly

The panel sub-assembly is the building block of the formwork at the back side of the T corner and at pilasters.

1. First model all the needed formwork objects (custom parts and/or beams).
2. Start the **Formwork sub-assembly wizard** tool.
3. Pick the center point of the object at the back of the wall as shown below. This will be used as the insertion point when this panel sub-assembly is inserted in an actual T corner.

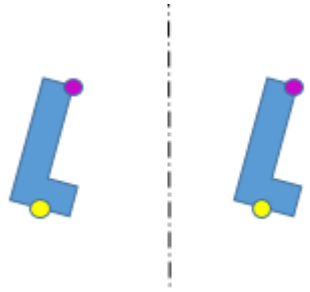
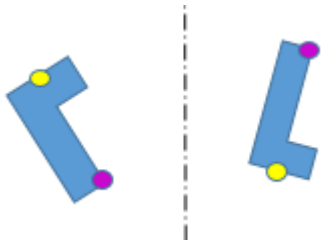
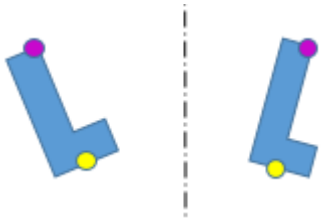
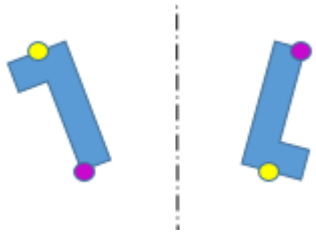


4. Pick a point to identify the direction.
5. Select the formwork parts (custom parts and/or beams) forming the panel sub-assembly.
6. Enter a name and click **Finish** to save the detailed panel and accessories into an external file.

Control the behavior of conditions with mirroring

When you define the corner sub-assemblies, it can be that the left-hand side corner needs to be placed in a right-hand side location. When this happens, the corner sub-assembly needs to be mirrored. Mirroring custom parts or formwork items is not always possible, especially when the part is not symmetric in any direction. The condition placing tool can use four different methods for mirroring. With the configuration file

`xxxxx.SubAssemblyItems.ini` you can control which method will be used for a certain corner sub-assembly case. The file contains the mirroring method keyword, and after the keyword you can introduce the names of the items or partial name tags to identify multiple matching items.

Mirroring method	Description
`Move` 	Default method. The custom part or formwork item is moved by an offset measured from the mirroring line to the center of the custom part or formwork item.
`RotateAroundZ` 	Input points are rotated 180 degrees around the center point at the mirroring line.
`RotateAroundAxis` 	Input points are first rotated 180 degrees around the mirroring line and then the custom part is rotated 180 degrees around the input axis.
`Mirror` 	Input point locations are mirrored and then the start and end points are swapped.
`FlipLocalXY`	Custom part or formwork item is moved and the local x and y axis at center are flipped (=rotated around local z 180 degrees).

Mirroring method	Description
FlipLocalXZ	Item or custom part is moved and local x and z axis at center are flipped (=rotated around local y 180 degrees).
FlipLocalYX	Item or custom part is moved and local y and z axis at center are flipped (=rotated around local x 180 degrees).
TurnUpSideDown	Item or custom part is moved and the mirroring is done by turning the component upside down in the global vertical direction.

An example of the content in a configuration xxxxx.SubAssemblyItems.ini file.

```
// This file allows you to define how items and custom parts in corner
sub assemblies
// are handled when placing the right handed corner into left handed
situation or vice versa.
//
// By default items are just moved and no true mirroring happens. In
following lines you can specify
// the mirroring methods used for certain items or custom parts. You can
introduce full names or partial names.
//
// The possible methods are:
//  #Mirror - input points are mirrored
//  #RotateAroundAxis - input points are mirrored and the item/custom
part is rotated 180 degrees around the axis:
//  #RotateAroundZ - input points are rotated 180 degrees around the
center point
//  #FlipLocalXY - the item/custom part is moved and local X and Y axis's
at center are flipped (=rotated around local Z 180 degrees)
//  #FlipLocalXZ - the item/custom part is moved and local X and Z axis's
at center are flipped (=rotated around local Y 180 degrees)
//  #FlipLocalYZ - the item/custom part is moved and local Y and Z axis's
at center are flipped (=rotated around local X 180 degrees)
//  #TurnUpSideDown - the item/custom part is moved and "mirroring" is
done by turning the component up-side.down
//
// Just list the names or partial names following the method keyword. Do
not change the key words.
//

#Mirror
_FIXING_BOLT

#RotateAroundAxis
DOKA-3D-583002000

#RotateAroundZ
_INSIDE
_OUTSIDE
_CORNER
_LEFT
_RIGHT

#FlipLocalXY

#FlipLocalXZ
```

#FlipLocalYX

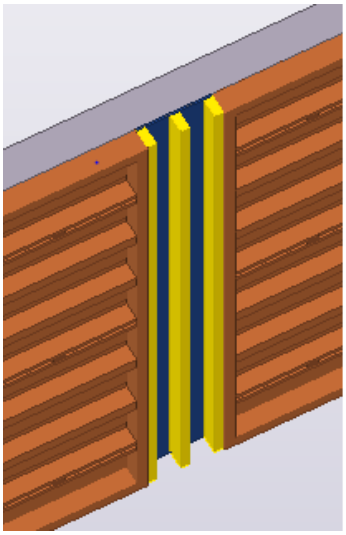
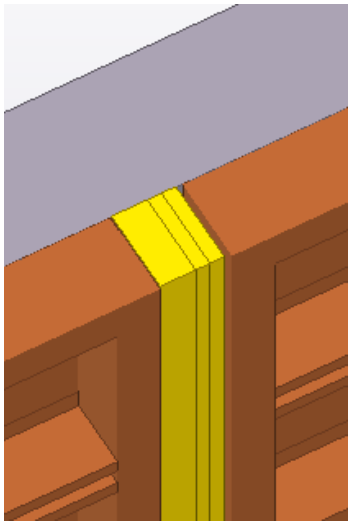
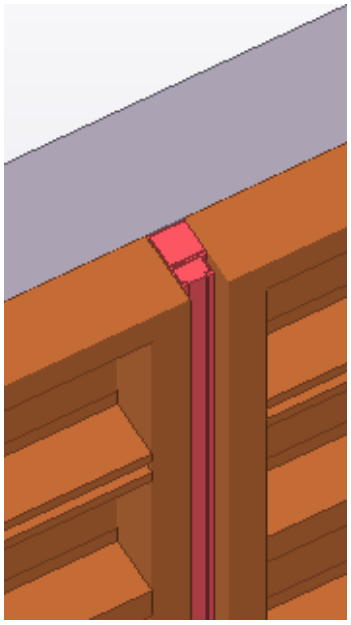
#TurnUpSideDown

Configure the fillers for the filler placing tool

xxxx.FormworkTools.Fillers.csv

With the filler placing tool you can create a filler by inserting timber beams or plywood and timber studs, and/or specific custom parts in the model. The creation of these model objects depends on the actual space to be filled. The basic principle of the creation of the various filler items is the following:

- If the space to be filled is larger than the given minimum plywood width, the filler tool will create the plywood (=contour plate) and the timber studs.
- If the space to be filled is smaller than the minimum plywood width, or plywood is not specified, the actual space is filled by one or more timber beams or custom parts. The target is to fill the actual space as completely as possible. The largest beams and/or custom parts are preferred and smaller pieces are created only when the space is smaller than the size of the larger pieces.

Filler built with plywood and studs	Filler built with timber beams	Filler built with custom part
		

To configure the filler tool, you can specify the properties for timber beam, plywood, and settings for one to ten alternatives of custom parts.

Definition	Description
Supplier	Supplier name, which is typically the same for all rows. The name is shown in the **Formwork supplier** list.
Family	Family name, which is shown in the **Product family** list.
Name	Unique filler name.
TimberWidths	List of available timber beam widths (10 20 50 100).
TimberDepth	Depth of the timber beam.
TimberAssPrefix	Assembly prefix.
TimberAssStartNo	Assembly start number.
TimberPartPrefix	Part prefix.
TimberPartStartNo	Part start number.
TimberName	Name of the timber beam.
TimberClass	Class of the timber beam.
TimberFinish	Finish of the timber beam.
TimberMaterial	Material of the timber beam.
StudWidth	Width of the timber stud.
StudDepth	Depth of the timber stud.
StudAssPrefix	Assembly prefix.
StudAssStartNo	Assembly start number.
StudPartPrefix	Part prefix.
StudPartStartNo	Part start number.
StudName	Name of the timber stud.
StudClass	Class of the stud.
StudFinish	Finish of the stud.
StudMaterial	Material of the stud.
PlywoodWidth	Minimum width of the plywood. The actual width depends on the actual space to be filled.
PlywoodThickness	Thickness of the plywood.
PlywoodAssPrefix	Assembly prefix.
PlywoodAssStartNo	Assembly start number.
PlywoodPartPrefix	Part prefix.
PlywoodPartStartNo	Part start number.
PlywoodName	Name of the plywood part.

Definition	Description
`PlywoodClass`	Class of the plywood part.
`PlywoodFinish`	Finish of the plywood part.
`PlywoodMaterial`	Material of the plywood part.
`Part1Name`	Name of the custom part 1.
`Part1Attributes`	Property file saved in the component dialog box to be used when the custom part is inserted in the model.
`Part1PlanePosition`	This is **Position in plane** when the custom part is inserted in the face view of the main formwork panel.
`Part1PlaneOffset`	This is **Offset in plane** when the custom part is inserted in the face view of the main formwork panel.
`Part1Rotation`	This is **Rotation** when the custom part is inserted in the face view of the main formwork panel.
`Part1RotationOffset`	This is **Rotation offset** when the custom part is inserted in the face view of the main formwork panel.
`Part1DepthPosition`	This is **Position in depth** when the custom part is inserted in the face view of the main formwork panel.
`Part1DepthOffset`	This is **Offset in depth** when the custom part is inserted in the face view of the main formwork panel.
`Part1StartOffset`	Offset of the first insertion point in relation to **Input point 1**.
`Part1EndOffset`	Offset of the first insertion point in relation to **Input point 2**.
`Part1Length`	Length of the custom part which is also the space to fill if/when this custom part is added to the model.
`Part1Height`	Height of the custom part.
`Part1LengthProperty`	Name of the length property of the custom part, if you are using a custom part whose length/width can be can be varying and/or parametric. The fields can be set also for `Part2`, `Part3`, and so on. You can configure multiple parts with separate additional rows. You need to keep the

Definition	Description
	`Supplier`, `Family`, and `Name` the same as on the first row.
`Part1LengthMin`	Minimum length/width (the free gap to fill) the product can fill, if you are using a custom part whose length/width can be can be varying and/or parametric. The fields can be set also for `Part2`, `Part3`, and so on. You can configure multiple parts with separate additional rows. You need to keep the `Supplier`, `Family`, and `Name` the same as on the first row.
`Part1LengthMax`	Maximum length/width the product can fill, if you are using a custom part whose length/width can be can be varying and/or parametric. The fields can be set also for `Part2`, `Part3`, and so on. You can configure multiple parts with separate additional rows. You need to keep the `Supplier`, `Family`, and `Name` the same as on the first row.
`Part1HeightProperty`	Name of the height property of the custom part, if you are using a custom part whose length/width can be can be varying and/or parametric. The fields can be set also for `Part2`, `Part3`, and so on. You can configure multiple parts with separate additional rows. You need to keep the `Supplier`, `Family`, and `Name` the same as on the first row.
`Part2... Part3... Part10...`	In addition to the custom part 1, you can specify up to nine other custom parts. In practice, at least the `PartXLength` needs to be different for all specified custom parts. The tool selects the part with the most suitable length depending on the actual space to be filled.

Formwork placing tools - Slabs

Formwork placing tools - Slabs is a set of components that helps in detailed modeling of typical slab formwork. These components are placing tools, and therefore you need to have all relevant formwork products such as panels, girders, and shuttering props available in the **Applications & components** catalog.

You can get the formwork products provided by formwork suppliers from **Tekla Warehouse**, for example. In addition, **Formwork placing tools - Slabs** requires additional configuration files that contain the necessary information about the formwork product components. These configuration files may come with the formwork product catalogs provided by formwork suppliers, but you can create the configuration files yourself as well.

Formwork placing tools - Slabs contains tools to place and modify the following formwork elements:

- formwork slab panels typically appearing as two-dimensional arrays covering a certain area
- cross and main girders supporting the slab panels
- shuttering props supporting the girders or slab panels directly depending on the formwork system
- stop ends on top of slab panels
- safety railings in the outer edges of the slab panel area

Because **Formwork placing tools - Slabs** consists of a set of sub-tools which are combined in one dialog box, each tab is an individual sub-tool. Both the **Formwork placing tools - Slabs** and the sub-tools are available in **Applications and components** catalog.

Most of the settings in the components are preset. You can control the different settings by selecting a suitable options in the dialog box. These preset settings are organized according to the formwork supplier and the product families.

However, if you create your own configuration files, you can use the values in the configuration files instead of the preset values.

You can access the **Formwork placing tools - Slabs** in the **Concrete Contractor**, **General Contractor** and **Rebar Detailer** roles in the **Default** environment.

Panel tab

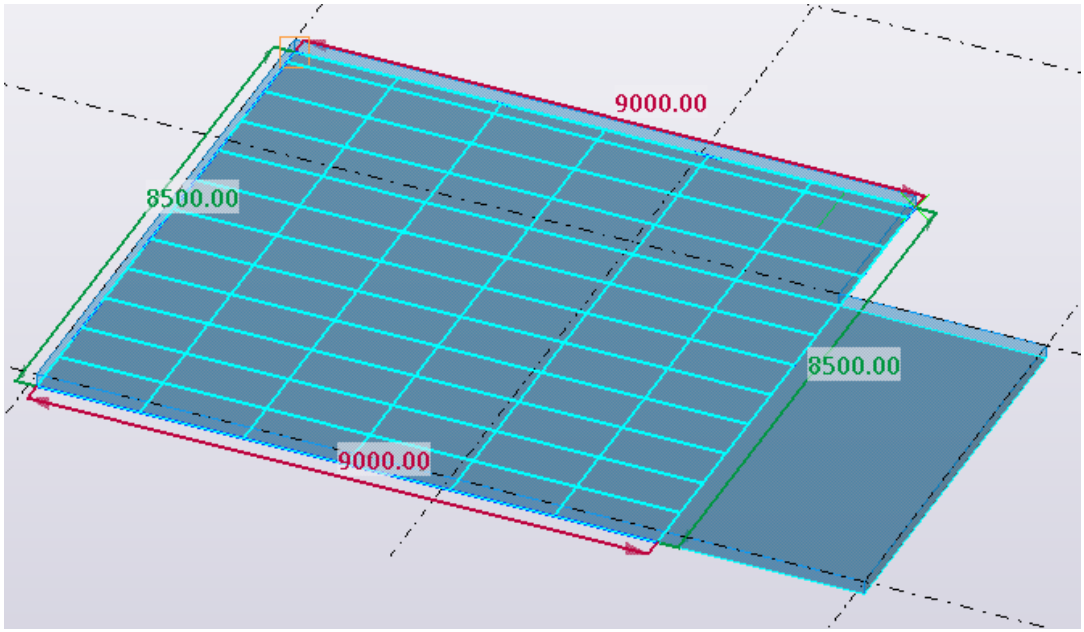
Use the **Panel** tab to model an area of formwork panels for slabs.

1. On the **Panel** tab, in the **Formwork supplier** and **Product family** list, select the suitable options.

   The options vary depending on which catalogs and tool configurations you have in use.

2. Select the panel size in the **Formwork panel** list, and set the other panel properties.
3. Click **Apply** and **Insert new** to start placing the slab panels.
4. Hover the mouse pointer on top of the slab to identify the bottom surface.
5. Select the slab and pick three or more points to identify the area on the surface. A preview of the panels is shown while you pick the points.

   Alternatively, hold down the **Alt** key and select the surface to create panels on the whole surface.
6. To finish, click the middle mouse button.
7. If you need to modify the panel, drag the corner points.



Cross girders and Main girders tabs

Use the **Cross girders** and **Main girders** tabs to create cross and main girders in two directions, or only main girders, or no girders at all.

1. On the **Cross girders** or on the **Main girders** tab, select the girder in the **Girder name** list.
2. Enter an **Overlap** value. If you leave the value empty, the tool sets it automatically.
3. Enter the girder **Length**, if the girder length is not fixed. Use space as a separator if you need to have girders with different lengths.
   a. To create girders at once, select the panel area, and click **Apply** and **Add to selected**.

      The **Add shoring** dialog box opens.

b. Set the girder offset and spacing.

   You can also select the option whether to create girders at the panel area edges, and the girder layout at the panel seam.

c. Click **Create**.

a. To create a single line of girders, click **Apply** and **Insert new**.

b. Pick the girder points.

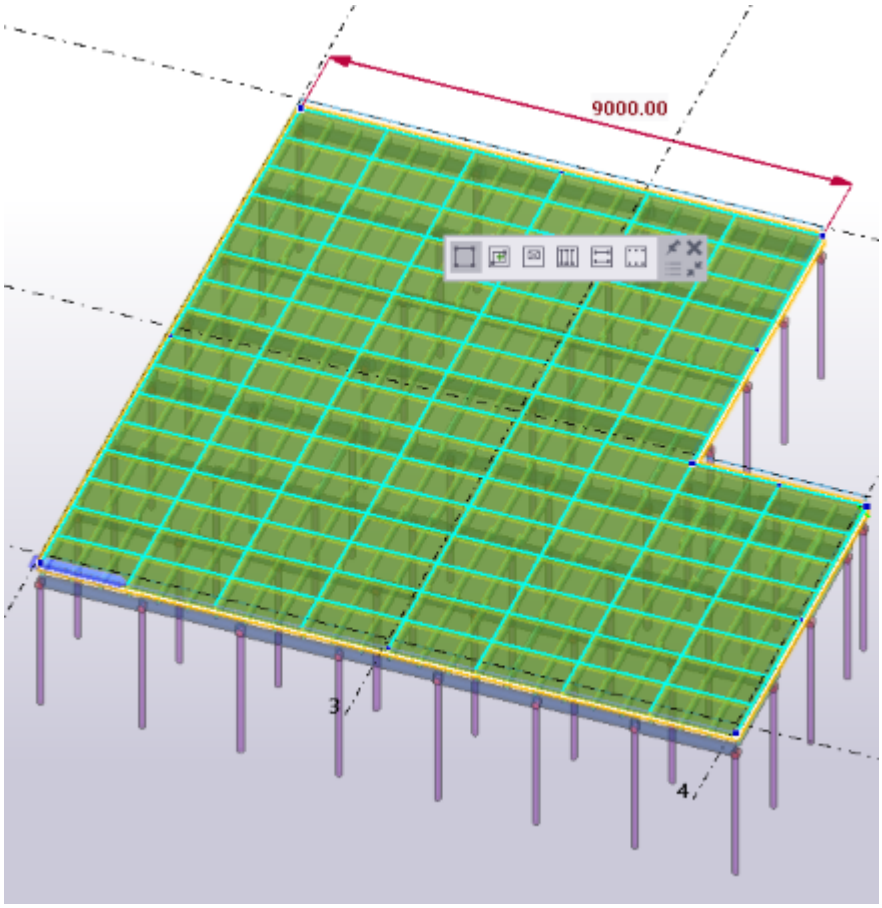
c. To finish, click the middle mouse button.

Shuttering tab

Use the **Shuttering** tab to create shuttering props, shuttering bases, and shuttering heads.

1. On the **Shuttering** tab, select the prop name in the **Girder name** list, the prop head name in the **Head name** list, and the prop base name in the **Base name** list.
2. Enter the total height for the prop.
   - To create multiple shuttering props, select the panel area and click **Apply** and **Add to selected**.

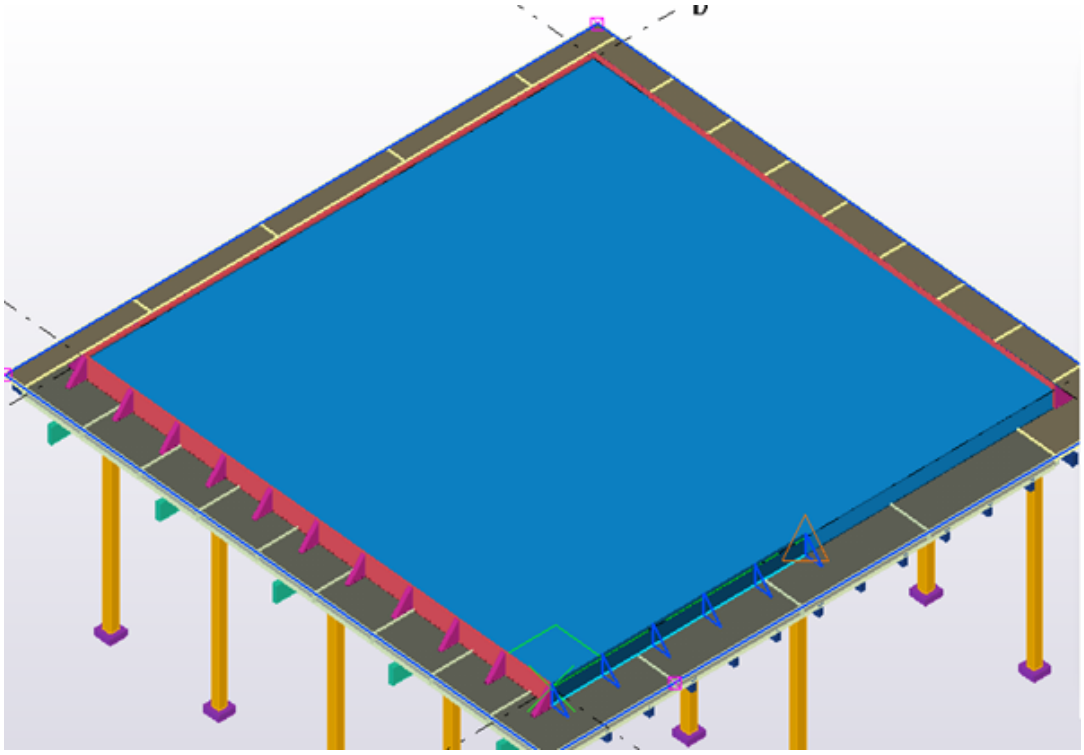
     The shuttering props are created.
   - To create a single shuttering prop, click **Apply** and **Insert new**. Pick the shuttering prop location.



Stop ends tab

Use the **Stop ends** tab to create stop ends around the slab panel.

1. On the **Stop ends** tab, select the stop end name in the **Stop end name** list.
2. Enter the offset and spacing for the stop ends.
3. Click **Apply** and **Insert new** to start inserting the stop ends.
   - If no panel is selected, first select the slab panel.
   - If you have already selected the slab panel, you do not need to do it again.
4. Pick the start point and the end point for the stop end.
5. To finish, click the middle mouse button.



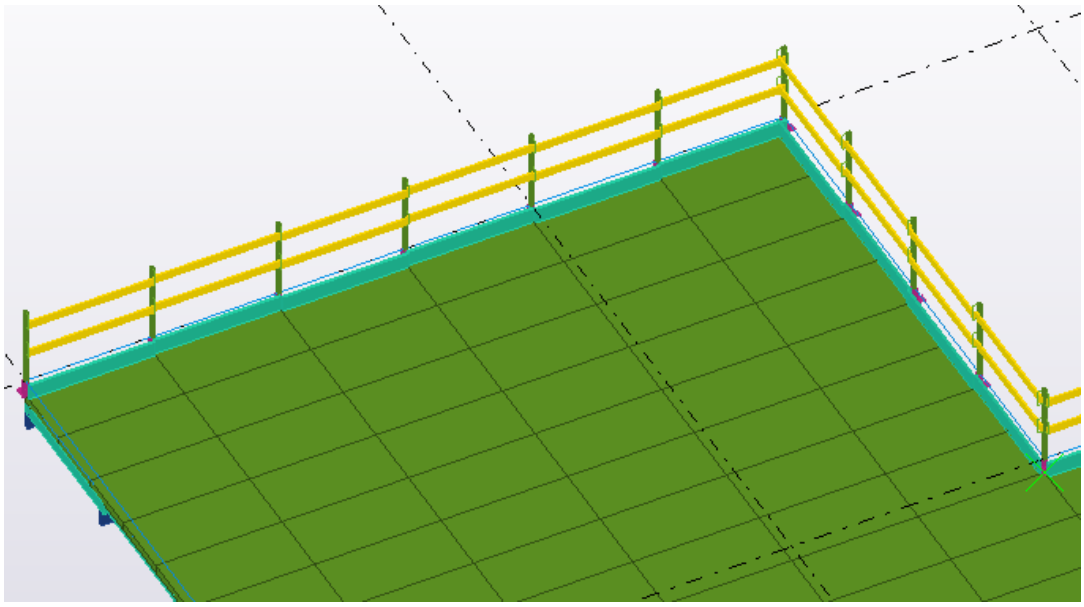
Railings tab

Use the **Railings** tab to create railings around the slab.

1. On the **Railings** tab, select the railing name in the **Railing name** list.
2. Enter the offset and spacing for the railings.
3. Click **Apply** and **Insert new** to start inserting the railings.
4. • If the panel is selected, first select the slab panel.

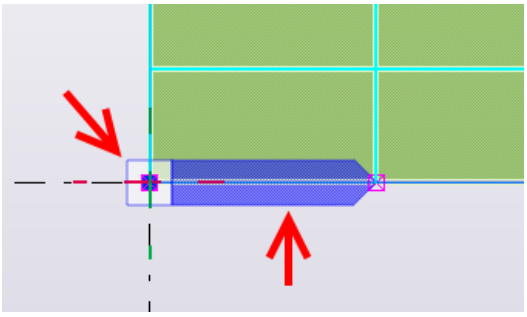
   Depending on the railing type, you may need to select the top face of the slab, side face of the wall, or the slab panels.

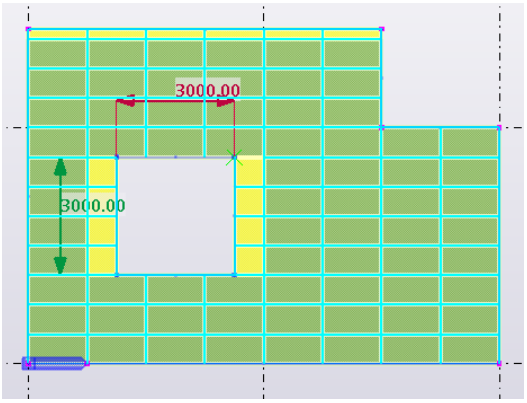
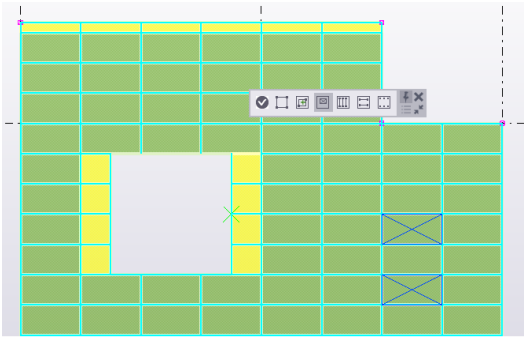
   • If you have already selected the slab panel, you do not need to do it again.
5. Pick the start point and the end point for the railing.
6. To finish picking, click the middle mouse button.

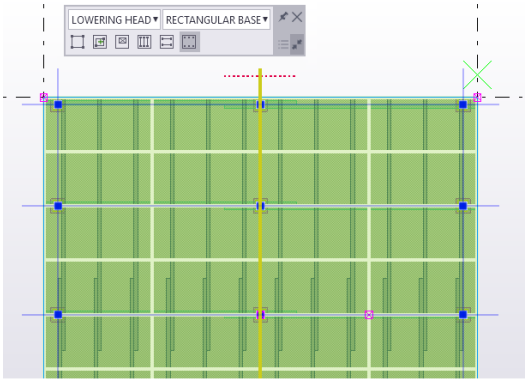


Contextual toolbar for formwork slab panel

When you select a slab panel, the following contextual toolbar appears and you can modify the parts:

Option	Description
	Modify the boundary of the panel area and openings. Move edges or points by dragging, or insert new points by dragging the insertion points at the middle of the edges. In addition, you can move the start point, which is the corner of the panel, and direction of the panel by dragging the handles. 

Option	Description
	Modify the panel width and length in the panel array.
	Add new rectangular openings in the panel area. After adding a rectangular opening, you can edit the opening to any shape. Typically, the openings are larger areas where the panels are not needed. To replace the regular panel with some special infill parts, use the single panel removal tool . 
	Remove or restore single panels in the area. Select the panels to be removed and click . Select multiple panels by holding down the **Shift** key when clicking the panels. Hold down the **Ctrl** key to unselect the panels. 
	Modify cross girders. To modify a single girder line, select the line to modify it.

Option	Description
	Modify main girders. To modify a single girder line, select the line to modify it.
	Modify shuttering props in the panel area. You can • move the individual props by dragging the handles. Hold down the **Ctrl** key to create a copy of the prop. • move a whole line of props by dragging the line handles. Hold down the **Ctrl** key to create a copy of all the props in the line. • change the shuttering prop head or base for shuttering props on all selected points and/or lines. Note that props created for the girder lines can only be moved/copied along the girder line. 

Formwork placing tools - Slabs: configuration

You can set up your own configuration files for **Formwork placings tools - Slabs**.

The configuration is done by using comma separated files (`.csv`), which can be edited with Microsoft Excel or any standard text editor. Each separate formwork sub-tool component has its own configuration file.

The configuration files can be located in any of the system folders, or in the sub-folder called `Formwork tools` in the current model folder.

Example configuration files are located in the `...\ProgramData\Trimble\Tekla Structures\<version>\Environments\common\system\CIP\Formwork` folder.

The configuration files are typically named by the formwork supplier and/or the product families. There can be any number of files, and the files are identified with a specific suffix. Each of the files controls one of the sub-tools in the component. The files contain a varying number of columns.

Use the following configuration files for **Formwork placings tools - Slabs** to set up

- panels: `xxxx.FormworkTools.SlabPanels.csv`
- girders: `xxxx.FormworkTools.Girders.csv`
- prop parts of shuttering props: `xxxx.FormworkTools.Props.csv`
- prop bases of shuttering props: `xxxx.FormworkTools.PropBases.csv`
- prop heads of shuttering props: `xxxx.FormworkTools.PropHeads.csv`
- stop ends: `xxxx.FormworkTools.StopEnds.csv`
- safety railings: `xxxx.FormworkTools.Railings.csv`

Accessories in formwork placing tools

You can set up any formwork accessory to be created as a beam, an item, or as a custom part. For beams and items you can additionally set up any of the beam or items properties or UDAs either in the configuration file by adding new columns, or as property files.

Each configuration file contains a header row and data rows. The header row is the first row that is not a comment row, and it gives the configuration parameter names (column name). Each data row gives one product and defines the parameters used when placing the beam, item or custom part.

In addition to header row and data rows the file can contain comment rows. A comment row is any row starting with text //.

You can define the distance unit by adding a row:

```
DISTANCE_UNIT=MM
```

When the distance unit is defined using the above setting, all distance values can be given as decimal values in the specified units. The supported units are: `MM`, `DN`, `CM`, `M`, `INCH`, `FEET`.

The following accessories are available for different **Formwork placings tools - Slabs** product types.

Product type	Accessory
Panel	`Panel` `Part`

Product type	Accessory
Girder	`Girder` `Beam`
Prop	`Prop`
Base	`Base`
Head	`Head`
Stop end	`Sheet` `Angle`

Railing	`Post`
	`Shoe`
	`ToeBoard`
	`GuardRail`

Common configuration parameter fields

To configure the accessories in any of the formwork product types, define the following fields in the `.csv` configuration file.

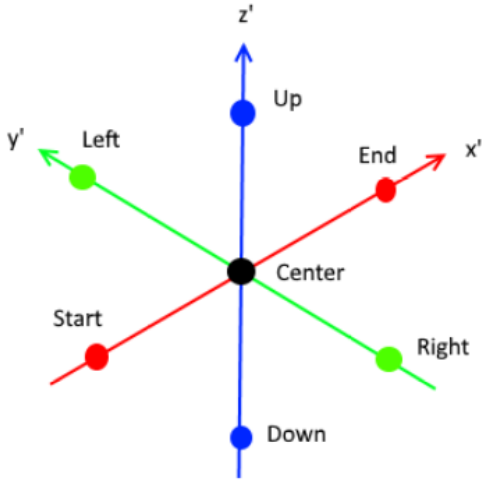
The text `[Accessory]` in the following table denotes the accessory in question.

Column name in the .csv file	Description

`[Accessory].Profile`
`[Accessory].Item`
`[Accessory].CustomPart`
	Formwork placing tools can place either beams, items, or custom parts to the model.
	If all products are of the same model object type you can use one of the three field names and give the corresponding data value.
	The field name specifies the model object default type but you can also override the model object type by adding the prefix `BEAM:`, `ITEM:` or `CUSTOMPART:` for the actual value.
	The actual value for `BEAM:` is a valid profile string, for `ITEM:` a valid item shape name, and for `CUSTOMPART:` a valid custom part name.

`[Accessory].Attributes`	Saved property file used to set either part, item or custom part properties.
`[Accessory].Name`	Name of the part or the item, or the **Name** property of the custom part.

Column name in the .csv file	Description
`[Accessory].Class`	Class of the part or the item, or the **Class** property of the custom part.
`[Accessory].Material`	Material of the part or the item, or the **Material** property of the custom part.
`[Accessory].AssPrefix`	Part or item numbering assembly prefix, or the **AssPrefix** property of the custom part.
`[Accessory].AssStartNo`	Part or item assembly start number, or the **AssStartNo** property of the custom part.
`[Accessory].PartPrefix`	Part or item numbering part prefix, or the **PartPrefix** property of the custom part.
`[Accessory].PartStartNo`	Part or item part start number, or the **PartStartNo** property of the custom part.
`[Accessory].Finish`	Finish of the part or the item, or the **Finish** property of the custom part.
`[Accessory].InputOrder`	Set the final locations of the input points in relation to the default generic start and end points. The possible values are: • `StartEnd`: default if no other value is specified. • `EndStart`: reverses the default start/end points. • `CenterStart`: the first input point is at the center of the start/end point and the second point is at the start point. • `CenterEnd`: the first input point is at the center of the start/end point and the second point is at the end point. • `CenterLeft`: the first input point is at the center of the start/end point and the second point is at the left point. • `CenterRight`: the first input point is at the center of the start/end point and the second point is at the right point.

Column name in the .csv file	Description
	• `CenterUp`: the first input point is at the center of start/end point and the second point is up from the center point. • `CenterDown`: the first input point is at the center of the start/end point and the second point is down from the center point. Note that if the `.csv` file does not contain any value, the default value `StartEnd` is used. 
`[Accessory].PlanePosition`	This is **Position in plane** when the beam, item, or custom part is inserted in the plane view. The options are `MIDDLE`, `LEFT`, and `RIGHT`.
`[Accessory].PlaneOffset`	Offset on plane. The default value is 0.
`[Accessory].Rotation`	This is **Rotation** when the custom part is inserted in the plane view. The options are `FRONT`, `TOP`, `BACK`, and `BELOW`.
`[Accessory].RotationOffset`	Rotation offset in degrees. The default value is 0.

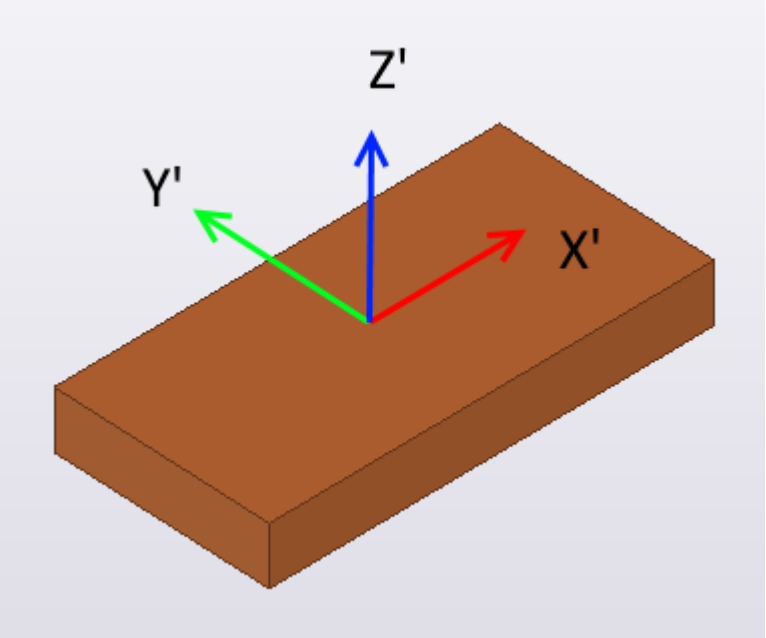
Column name in the .csv file	Description
[Accessory].DepthPosition	This is **Position at depth** when the beam, item, or custom part is inserted in the plane view. The options are MIDDLE, FRONT, and BEHIND.
[Accessory].DepthOffset	Offset at depth. The default value is 0.
[Accessory].StartOffset	Offset of the actual start point from the generic start point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
[Accessory].EndOffset	Offset of the actual end point from the generic end point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
[Accessory].UDA.XXXXXX	Define additional UDA values applied to the model objects. You can introduce as many UDA values as needed. Note that the UDA name (XXXXX) must be the internal name, not the localized name shown in the user interface.

Configure the slab panel placing tool

.FormworkTools.SlabPanels.csv

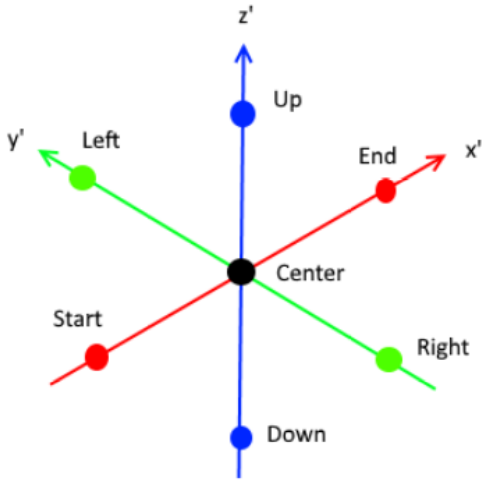
Add one row for each panel. Specify columns as follows. For a slab panel, you can specify one custom part that is placed if the panel is a full panel, and part properties used for placing a filler part at locations where the panel is not a full panel.

The configuration is done in the local coordinate system of the panel. The origin of the panel is located at the center point of the top of the panel.



Row	Definition
`Supplier`	Supplier name, this is typically the same for all rows. The name is shown in the **Formwork supplier** list.
`Family`	Family name, which is shown in the **Product family** list.
`Name`	Unique panel name, which is shown in the **Formwork panel** list.
`PanelName`	Name of the custom part that will be inserted in the model. This is the custom part that the panel configuration will use.
`PanelAttributes`	Property file saved in the component dialog box to be used when the custom part is inserted into model.
`PanelInputOrder`	Define the orientation of the custom part panels by specifying where the start and end handles of the custom part will go. Set the final locations of the input points in relation to the default generic start and end points. The possible values are:

Row	Definition
	• `StartEnd`: default if no other value is specified. • `EndStart`: reverses the default start/end points. • `CenterStart`: the first input point is at the center of the start/end point and the second point is at the start point. • `CenterEnd`: the first input point is at the center of the start/end point and the second point is at the end point. • `CenterLeft`: the first input point is at the center of the start/end point and the second point is at the left point. • `CenterRight`: the first input point is at the center of the start/end point and the second point is at the right point. • `CenterUp`: the first input point is at the center of start/end point and the second point is up from the center point. • `CenterDown`: the first input point is at the center of the start/end point and the second point is down from the center point. Note that if the `.csv` file does not contain any value, the default value `StartEnd` is used.

Row	Definition
	
`PanelPlanePosition`	Use the following position values to define the custom part's position, offset and rotation in relation to the custom part start and end points. Use **Position in plane** to insert the custom part in the plane view. The options are **MIDDLE**, **LEFT** and **RIGHT**.
`PanelPlaneOffset`	Offset on plane. The default value is 0.
`PanelRotation`	Use **Rotation** to insert the custom part in the plane view. The options are **FRONT**, **TOP**, **BACK**, and **BELOW**.
`PanelRotationOffset`	Rotation offset in degrees. The default value is 0.
`PanelDepthPosition`	Use **Position at depth** to insert the custom part in the plane view. The options are **MIDDLE**, **FRONT**, and **BEHIND**.
`PanelDepthOffset`	Offset at depth. The default value is 0.
`PanelStartOffset`	`PanelStartOffset` and `PanelEndOffset` change the location of the actual start and end handles of the custom part. `PanelStartOffset` is the offset of the actual start point from the generic start point in the local coordinate

Row	Definition
	system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
`PanelEndOffset`	`PanelEndOffset` is the offset of the actual end point from the generic end point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
`LengthProperty`	The name of the length property in the custom part. If the length is fixed, this is empty.
`LengthValue`	Length of the panel. Note that the length value is read from the `.csv` file.
`WidthProperty`	The name of the width property in the custom part. If the width is fixed, this is empty.
`WidthValue`	Width of the panel. Note that the width value is read from the `.csv` file.
`ThicknessProperty`	The name of the thickness property in the custom part. If the thickness is fixed, this is empty.
`ThicknessValue`	Thickness of the panel. Note that the thickness value is read from the `.csv` file.
`CrossGirderX`	The girder values control the default spacing of the girders visible in the **Add shoring** dialog box. The values in the `.csv` file are taken from the center of the panel. `CrossGirderX` specifies the default x locations of the cross girders. Values must be enclosed in parentheses () and separated by spaces.
`MainGirderY`	`MainGirderY` specifies the default y locations of the main girders. Values

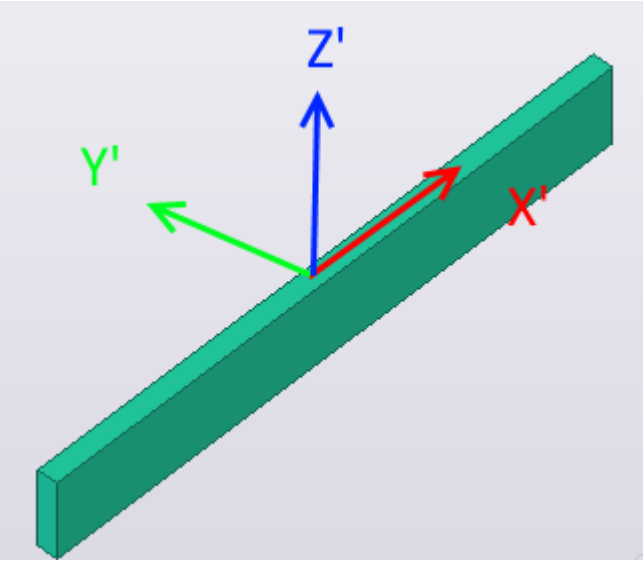
Row	Definition
	must be enclosed in parentheses () and separated by spaces.
PartName	The following part properties refer to the properties of the defined filler parts used when the custom parts cannot fill the whole slab area. Name of the panel part (contour plate).
PartClass	Class of the part.
PartFinish	Finish of the part.
PartMaterial	Material of the part.
PartThickness	Thickness of the part.
PartAssPrefix	Assembly prefix.
PartAssStartNo	Assembly start number.
PartPartPrefix	Part prefix.
PartPartStartNo	Part start number.

Configure the girder line tool

`.xxxx.FormworkTools.Girders.csv`

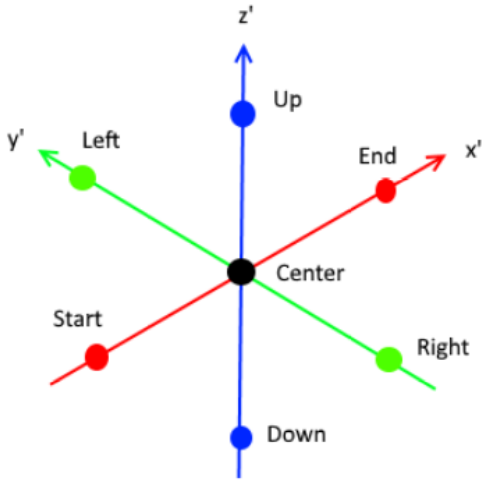
For a girder, you can specify either a custom part or a beam which is placed in the model.

The position settings are given in the local coordinate system of the girder.



Row	Definition
`Supplier`	Supplier name, which is typically the same for all rows. The name is shown in the **Formwork supplier** list.
`Family`	Family name, which is shown in the **Product family** list.
`Name`	Girder name, which is shown in the **Girder name** list. This name can be unique, or you can specify two or more girders with the same name. If one girder name has two or more specifications, the placing tool may create a line containing girders with variable lengths.
`Type`	The type of the girder. The type defines the list in which the girder is shown. The options are: `CROSS`, `MAIN` and `ANY`.
`Length`	Length (x) of the girder. Note that the length value is read from the `.csv` file.
`Width`	Width (y) of the girder. Note that the value is read from the `.csv` file.
`Depth`	Depth (z) of the girder. Note that the depth value is read from the `.csv` file.
`MinLength`	If the length varies, this is the minimum length of the girder.
`MaxLength`	If the length varies, this is the maximum length of the girder.
`MinOverlap`	When two or more girders are placed in one girder line, the girders can overlap to achieve the exact start and end for the line. The `MinOverlap` value controls the minimum overlap.
`MaxOverlap`	When two or more girders are placed in one girder line, the girders can overlap to achieve the exact start and end for the line. The `MaxOverlap` value controls the maximum overlap.

Row	Definition
	If the girders should not overlap, enter 0 as the value.
`InputPointLocation`	The depth location of girder input points. The options are: • `CrossGirderBottom` (only for main girders) • `SlabBottom` • `SlabTop`
`GirderName`	Name of the custom part that will be inserted in the model. This is the custom part that the girder configuration will use.
`GirderAttributes`	Property file saved in the component dialog box to be used when the custom part is inserted into model.
`GirderInputOrder`	Define the orientation of the custom part girders by specifying where the start and end handles of the custom part will go. Set the final locations of the input points in relation to the default generic start and end points. The possible values are: • `StartEnd`: default if no other value is specified. • `EndStart`: reverses the default start/end points. • `CenterStart`: the first input point is at the center of the start/end point and the second point is at the start point. • `CenterEnd`: the first input point is at the center of the start/end point and the second point is at the end point. • `CenterLeft`: the first input point is at the center of the start/end point and the second point is at the left point. • `CenterRight`: the first input point is at the center of the start/end

Row	Definition
	point and the second point is at the right point. • `CenterUp`: the first input point is at the center of start/end point and the second point is up from the center point. • `CenterDown`: the first input point is at the center of the start/end point and the second point is down from the center point. Note that if the `.csv` file does not contain any value, the default value `StartEnd` is used. 
`GirderPlanePosition`	Use the following position values to define the custom part's position, offset and rotation in relation to the custom part start and end points. Use **Position in plane** to insert the custom part in the plane view. The options are **MIDDLE**, **LEFT**, and **RIGHT**.
`GirderPlaneOffset`	Offset on plane. The default value is 0.
`GirderRotation`	Use **Rotation** to insert the custom part in the plane view. The options are **FRONT**, **TOP**, **BACK**, and **BELOW**.

Row	Definition
GirderRotationOffset	Rotation offset in degrees. The default value is 0.
GirderDepthPosition	Use **Position at depth** to insert the custom part in the plane view. The options are **MIDDLE**, **FRONT**, and **BEHIND**.
GirderDepthOffset	Offset at depth. The default value is 0.
GirderStartOffset	GirderStartOffset and GirderEndOffset change the location of the actual start and end handle points of the custom part. GirderStartOffset is the offset of the actual start point from the generic start point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
GirderEndOffset	GirderEndOffset is the offset of the actual end point from the generic end point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
GirderLengthProperty	The name of the length property in the custom part. If the length is fixed, this is empty.
GirderWidthProperty	The name of the width property in the custom part. If the width is fixed, this is empty.
GirderDepthProperty	The name of the depth property in the custom part. If the width is fixed, this is empty.
BeamName	Name of the girder part (beam).
BeamProfile	Profile of the beam. If this is empty, the beam is not created.
BeamClass	Class of the girder part.
BeamFinish	Finish of the girder part.
BeamMaterial	Material of the girder part.

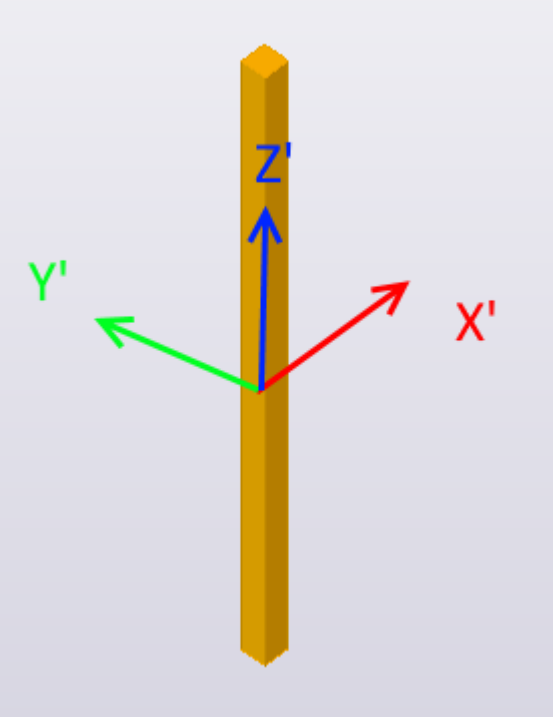
Row	Definition
BeamThickness	Thickness of the girder part.
BeamAssPrefix	Assembly prefix.
BeamAssStartNo	Assembly start number.
BeamPartPrefix	Part prefix.
BeamPartStartNo	Part start number.

Configure the shuttering prop placing tool

xxxx.FormworkTools.Props.csv

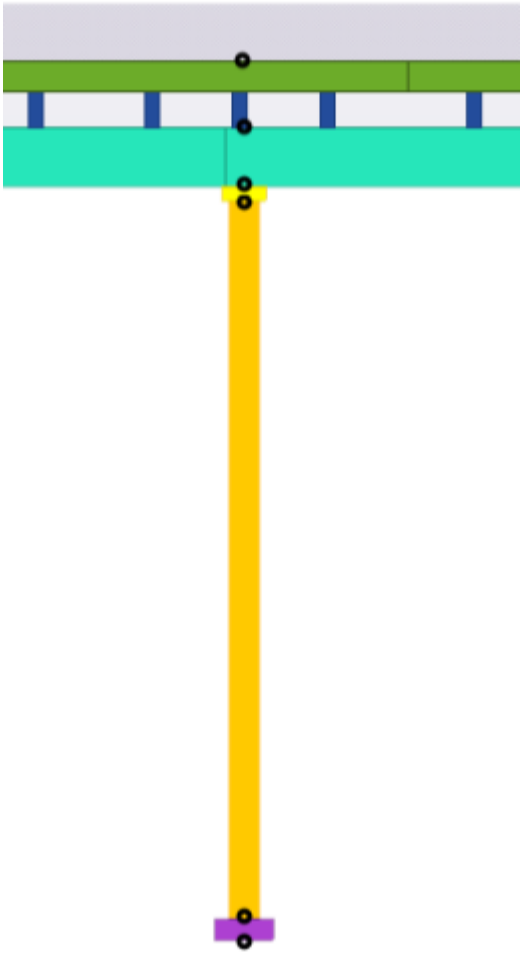
For a prop, you can specify a custom part that is placed in the model. In addition, a shuttering prop may contain the base and head parts, but these are configured in separate files (xxxx.FormworkTools.PropBases.csv and xxxx.FormworkTools.PropHeads.csv).

The position settings are given in the local coordinate system of the prop.

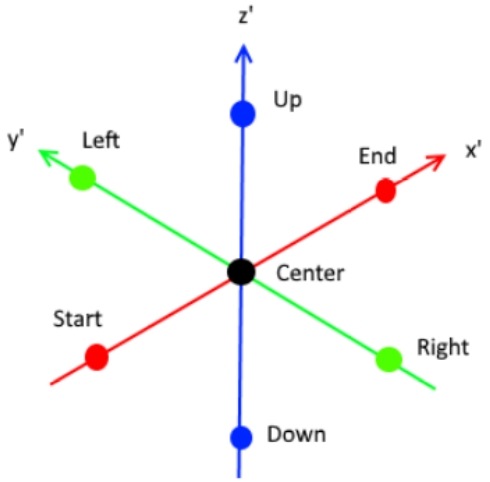


Row	Definition
Supplier	Supplier name, which is typically the same for all rows. The name is shown in the **Formwork supplier** list.
Family	Family name, which is shown in the **Product family** list.

Row	Definition
`Name`	Prop name, which is shown in the **Prop name** list. This name can be unique, or you can specify two or more props with the same name. If one prop name has two or more specifications, the placing tool can select the suitable prop based on the effective height of the shuttering.
`MinHeight`	The height of the prop from the bottom of the base prop to the bottom of the concrete slab. If the length varies, this is the minimum length. For a prop with fixed height the `MinHeight`and `MaxHeight` are same.
`MaxHeight`	If the length varies, this is the maximum length.
`PropInputPoint`	The location of the first input point of the custom part. The options are: • `BaseBottom` • `BaseTop` • `HeadBottom` • `GirderBottom` • `GirderTop` • `SlabBottom`

Row	Definition
	
`PropInputPoint2`	The location of the second input point of the custom part. If you use value `None`, the second input point is calculated as an offset `PropEndOffset` from the first input point.
`PropHeightProperty`	The name of the prop height property in the custom part. This is empty if the height is not parametric.
`PropName`	The name of the custom part that will be inserted in the model. This is the custom part that the prop configuration will use.
`PropAttributes`	Property file saved in the component dialog box to be used when the custom part is inserted into model.

Row	Definition
`PropInputOrder`	Define the orientation of the custom part prop by specifying where the start and end handles of the custom part will go. Set the final locations of the input points in relation to the default generic start and end points. The possible values are: • `StartEnd`: default if no other value is specified. • `EndStart`: reverses the default start/end points. • `CenterStart`: the first input point is at the center of the start/end point and the second point is at the start point. • `CenterEnd`: the first input point is at the center of the start/end point and the second point is at the end point. • `CenterLeft`: the first input point is at the center of the start/end point and the second point is at the left point. • `CenterRight`: the first input point is at the center of the start/end point and the second point is at the right point. • `CenterUp`: the first input point is at the center of start/end point and the second point is up from the center point. • `CenterDown`: the first input point is at the center of the start/end point and the second point is down from the center point. Note that if the `.csv` file does not contain any value, the default value `StartEnd` is used.

Row	Definition
	
`PropPlanePosition`	Use the following position values to define the custom part's position, offset and rotation in relation to the custom part start and end points. Use **Position in plane** to insert the custom part in the plane view. The options are **MIDDLE**, **LEFT**, and **RIGHT**.
`PropPlaneOffset`	Offset on plane. The default value is 0.
`PropRotation`	Use **Rotation** to insert the custom part in the plane view. The options are **FRONT**, **TOP**, **BACK**, and **BELOW**.
`PropRotationOffset`	Rotation offset in degrees. The default value is 0.
`PropDepthPosition`	Use **Position at depth** to insert the custom part in the plane view. The options are **MIDDLE**, **FRONT**, and **BEHIND**.
`PropDepthOffset`	Offset at depth. The default value is 0.
`PropStartOffset`	`PropStartOffset` and `PropEndOffset` change the location of the actual start and end handle points of the custom part. `PropStartOffset` is the offset of the actual start point from the generic start point in the local coordinate

Row	Definition
	system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
`PropEndOffset`	`PanelEndOffset` is the offset of the actual end point from the generic end point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used. If the `PropEndOffset` value is not given, or the value `None` is used, `PropEndOffset` is an offset from the first input point to the second input point.

Configure the base parts of the shuttering prop placing tool

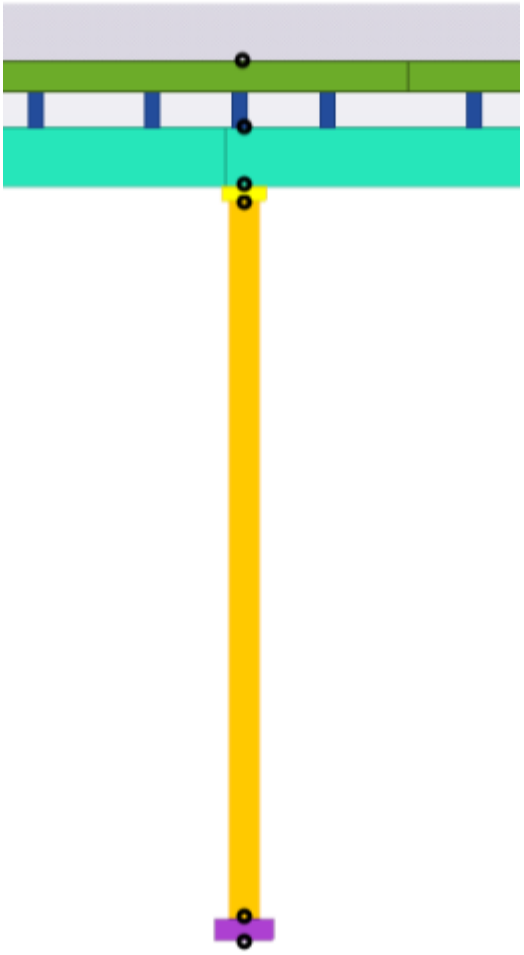
`xxxx.FormworkTools.PropBases.csv`

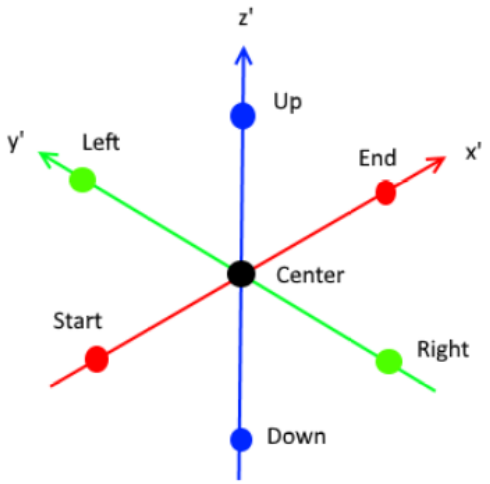
For a shuttering prop, you can specify an additional custom part that is placed in the model at the bottom end of the prop.

The position settings are given in the local coordinate system of the prop.

Row	Definition
`Supplier`	Supplier name, which is typically the same for all rows. The name is shown in the **Formwork supplier** list.
`Family`	Family name, which is shown in the **Product family** list.
`Name`	Unique prop base name, which is shown in the **Base name** list.
`PropInputPoint`	The location of the first input point of the custom part. The options are: • `BaseBottom` • `BaseTop` • `HeadBottom` • `GirderBottom` • `GirderTop`

Row	Definition
	• SlabBottom
PropOffset	Automatic offset (shortening) of the prop if the base part is created.
BaseName	Name of the custom part that will be inserted in the model.
BaseAttributes	Property file saved in the component dialog box to be used when the custom part is inserted into model.
BaseInputPoint	The location of the first input point of the custom part. The options are: • BaseBottom • BaseTop • HeadBottom • GirderBottom • GirderTop • SlabBottom The second input point of the custom part is calculated as an offset BaseEndOffset from the first input point.

Row	Definition
	
`PropInputOrder`	Set the final locations of the input points in relation to the default generic start and end points. The possible values are: • `StartEnd`: default if no other value is specified. • `EndStart`: reverses the default start/end points. • `CenterStart`: the first input point is at the center of the start/end point and the second point is at the start point. • `CenterEnd`: the first input point is at the center of the start/end point and the second point is at the end point.

Row	Definition
	• `CenterLeft`: the first input point is at the center of the start/end point and the second point is at the left point. • `CenterRight`: the first input point is at the center of the start/end point and the second point is at the right point. • `CenterUp`: the first input point is at the center of start/end point and the second point is up from the center point. • `CenterDown`: the first input point is at the center of the start/end point and the second point is down from the center point. Note that if the `.csv` file does not contain any value, the default value `StartEnd` is used. 
`BasePlanePosition`	Use the following position values to define the custom part's position, offset and rotation in relation to the custom part start and end points. Use **Position in plane** to insert the custom part in the plane view. The options are **MIDDLE**, **LEFT**, and **RIGHT**.
`BasePlaneOffset`	Offset on plane. The default value is 0.

Row	Definition
BaseRotation	Use **Rotation** to insert the custom part in the plane view. The options are **FRONT**, **TOP**, **BACK**, and **BELOW**.
BaseRotationOffset	Rotation offset in degrees. The default value is 0.
BaseDepthPosition	Use **Position at depth** to insert the custom part in the plane view. The options are **MIDDLE**, **FRONT**, and **BEHIND**.
BaseDepthOffset	Offset at depth. The default value is 0.
BaseStartOffset	BaseStartOffset is the offset of the actual start point from the generic start point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
BaseEndOffset	BaseEndOffset specifies the location of the second input point of the custom part in relation to the first input point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, zero offset (100 0 0) is used.

Configure the head parts of the shuttering prop placing tool

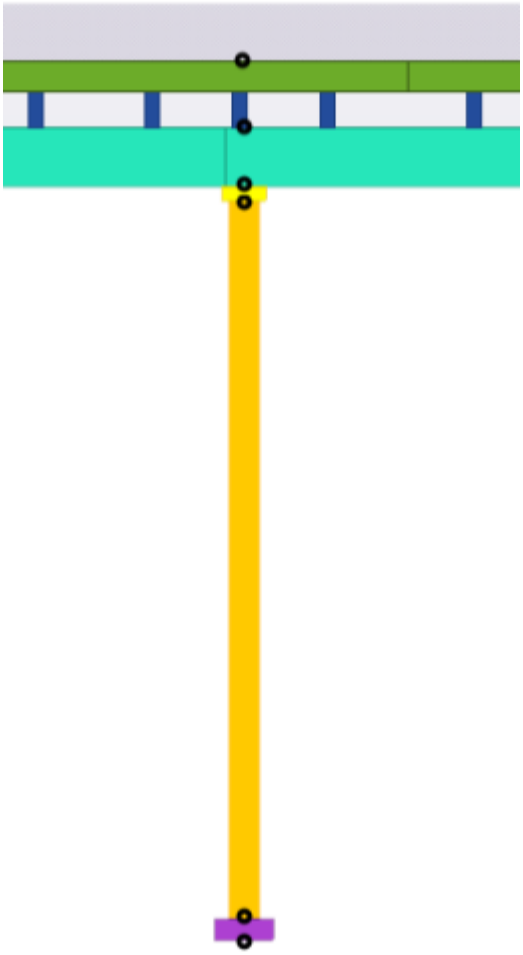
xxxx.FormworkTools.PropHeads.csv

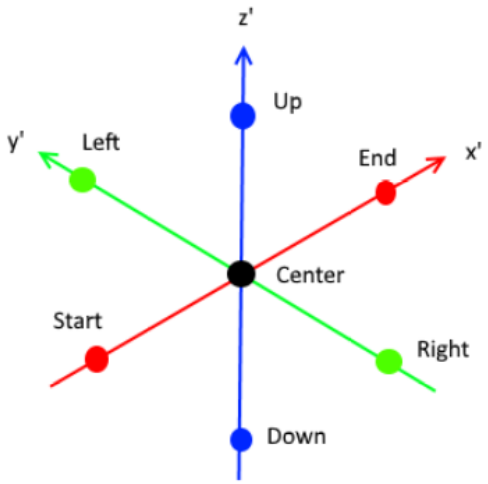
For a shuttering prop, you can specify an additional custom part that is placed in the model at the top end of the prop.

The position settings are given in the local coordinate system of the prop.

Row	Definition
Supplier	Supplier name, which is typically the same for all rows. The name is shown in the **Formwork supplier** list.
Family	Family name, which is shown in the **Product family** list.

Row	Definition
Name	Unique prop head name, which is shown in the **Head name** list.
PropOffset	Automatic offset (shortening) of the prop if the head part is created.
AutoRotate	Control whether the head part is rotated automatically 90 degrees around the prop axis when the head is located at the overlap of two girders. Set this value to YES to get the head rotated 90 degrees. Note that this only controls the rotation when you have not given any specific value for the rotation in the dialog box.
HeadName	Name of the custom part that will be inserted in the model.
HeadAttributes	Property file saved in the component dialog box to be used when the custom part is inserted into model.
HeadInputPoint	The location of the first input point of the custom part. The options are: • HeadBottom • HeadTop • HeadBottom • GirderBottom • GirderTop • SlabBottom The second input point of the custom part is calculated as an offset HeadEndOffset from the first input point.

Row	Definition
	
`HeadInputOrder`	Set the final locations of the input points in relation to the default generic start and end points. The possible values are: • `StartEnd`: default if no other value is specified. • `EndStart`: reverses the default start/end points. • `CenterStart`: the first input point is at the center of the start/end point and the second point is at the start point. • `CenterEnd`: the first input point is at the center of the start/end point and the second point is at the end point.

Row	Definition
	• `CenterLeft`: the first input point is at the center of the start/end point and the second point is at the left point. • `CenterRight`: the first input point is at the center of the start/end point and the second point is at the right point. • `CenterUp`: the first input point is at the center of start/end point and the second point is up from the center point. • `CenterDown`: the first input point is at the center of the start/end point and the second point is down from the center point. Note that if the `.csv` file does not contain any value, the default value `StartEnd` is used. 
`HeadPlanePosition`	Use the following position values to define the custom part's position, offset and rotation in relation to the custom part start and end points. Use **Position in plane** to insert the custom part in the plane view. The options are **MIDDLE**, **LEFT**, and **RIGHT**.
`HeadPlaneOffset`	Offset on plane. The default value is 0.

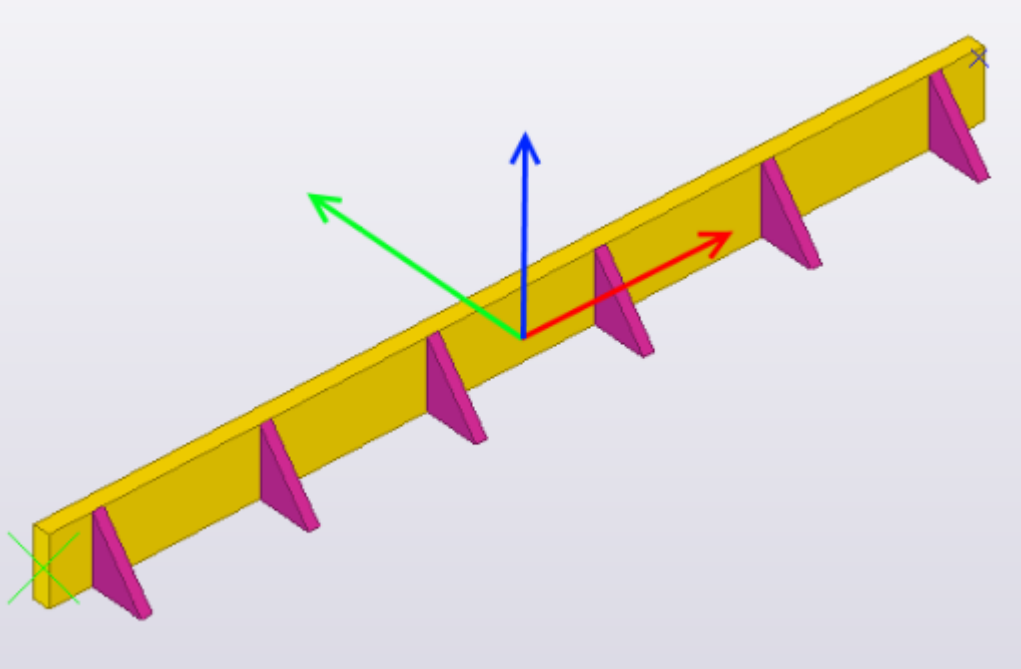
Row	Definition
`HeadRotation`	Use **Rotation** to insert the custom part in the plane view. The options are **FRONT**, **TOP**, **BACK**, and **BELOW**.
`HeadRotationOffset`	Rotation offset in degrees. The default value is 0.
`HeadDepthPosition`	Use **Position at depth** to insert the custom part in the plane view. The options are **MIDDLE**, **FRONT**, and **BEHIND**.
`HeadDepthOffset`	Offset at depth. The default value is 0.
`HeadStartOffset`	`HeadStartOffset` is the offset of the actual start point from the generic start point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
`HeadEndOffset`	`HeadEndOffset` specifies the location of the second input point of the custom part in relation to the first input point in the local coordinate system. The offset is given as x, y and z values separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, zero offset (100 0 0) is used.

Configure the stop end custom parts for the stop end placing tool

`xxxx.FormworkTools.StopEnds.csv`

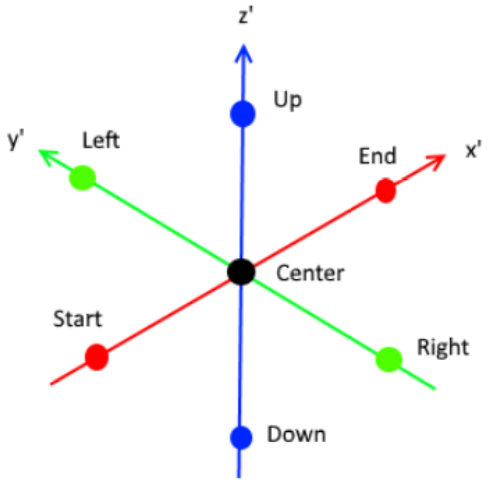
For a stop end, you can specify two custom parts: a sheet and an angle. The sheet is typically the actual form piece, and the angles are placed with a spacing to support the sheet.

The position settings are given in the local coordinate system of the stop end.

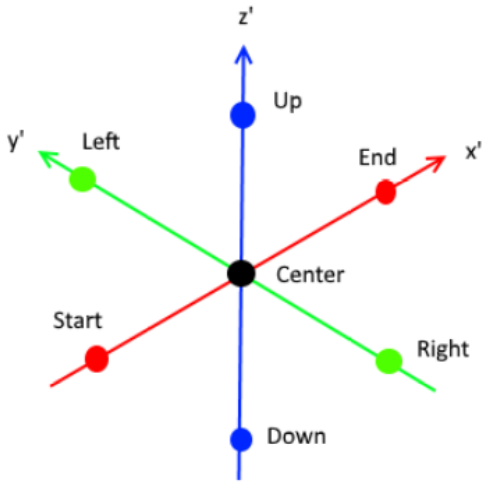


Row	Definition
`Supplier`	Supplier name, this is typically the same for all rows. The name is shown in the **Formwork supplier** list.
`Family`	Family name, which is shown in the **Product family** list.
`Name`	Unique stop end name, which is shown in the **Stop end name** list.
`MinLength`	Minimum length of the stop end sheet part.
`MaxLength`	Maximum length of the stop end sheet part.
`Width`	Total width of the stop end including the angles.
`Depth`	Total depth of the stop end.
`AngleSpacing`	Default spacing of the angle parts. This will be used if you do not specify any value for the spacing.
`SheetName`	Name of the custom part that will be inserted in the model.
`SheetAttributes`	Property file saved in the component dialog box to be used when the custom part is inserted into model.

Row	Definition
`SheetInputOrder`	Set the final locations of the input points in relation to the default generic start and end points. The possible values are: • `StartEnd`: default if no other value is specified. • `EndStart`: reverses the default start/end points. • `CenterStart`: the first input point is at the center of the start/end point and the second point is at the start point. • `CenterEnd`: the first input point is at the center of the start/end point and the second point is at the end point. • `CenterLeft`: the first input point is at the center of the start/end point and the second point is at the left point. • `CenterRight`: the first input point is at the center of the start/end point and the second point is at the right point. • `CenterUp`: the first input point is at the center of start/end point and the second point is up from the center point. • `CenterDown`: the first input point is at the center of the start/end point and the second point is down from the center point. Note that if the `.csv` file does not contain any value, the default value `StartEnd` is used.

Row	Definition
	
`SheetPlanePosition`	Use the following position values to define the custom part's position, offset and rotation in relation to the custom part start and end points. Use **Position in plane** to insert the custom part in the plane view. The options are **MIDDLE**, **LEFT**, and **RIGHT**.
`SheetPlaneOffset`	Offset on plane. The default value is 0.
`SheetRotation`	Use **Rotation** to insert the custom part in the plane view. The options are **FRONT**, **TOP**, **BACK**, and **BELOW**.
`SheetRotationOffset`	Rotation offset in degrees. The default value is 0.
`SheetDepthPosition`	Use **Position at depth** to insert the custom part in the plane view. The options are **MIDDLE**, **FRONT**, and **BEHIND**.
`SheetDepthOffset`	Offset at depth. The default value is 0.
`SheetStartOffset`	`SheetStartOffset` is the offset of the actual start point from the generic start point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.

Row	Definition
SheetEndOffset	SheetEndOffset specifies the location of the second input point of the custom part in relation to the first input point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, zero offset (100 0 0) is used.
AngleName	Name of the custom part that will be inserted in the model.
AngleInputOrder	Set the final locations of the input points in relation to the default generic start and end points. The possible values are: • StartEnd: default if no other value is specified. • EndStart: reverses the default start/end points. • CenterStart: the first input point is at the center of the start/end point and the second point is at the start point. • CenterEnd: the first input point is at the center of the start/end point and the second point is at the end point. • CenterLeft: the first input point is at the center of the start/end point and the second point is at the left point. • CenterRight: the first input point is at the center of the start/end point and the second point is at the right point. • CenterUp: the first input point is at the center of start/end point and the second point is up from the center point. • CenterDown: the first input point is at the center of the start/end

Row	Definition
	point and the second point is down from the center point. Note that if the `.csv` file does not contain any value, the default value `StartEnd` is used. 
`AnglePlanePosition`	Use the following position values to define the custom part's position, offset and rotation in relation to the custom part start and end points. Use **Position in plane** to insert the custom part in the plane view. The options are **MIDDLE**, **LEFT**, and **RIGHT**.
`AnglePlaneOffset`	Offset on plane. The default value is 0.
`AngleRotation`	Use **Rotation** to insert the custom part in the plane view. The options are **FRONT**, **TOP**, **BACK**, and **BELOW**.
`AngleRotationOffset`	Rotation offset in degrees. The default value is 0.
`AngleDepthPosition`	Use **Position at depth** to insert the custom part in the plane view. The options are **MIDDLE**, **FRONT**, and **BEHIND**.
`AngleDepthOffset`	Offset at depth. The default value is 0.
`AngleStartOffset`	`AngleStartOffset` is the offset of the actual start point from the generic

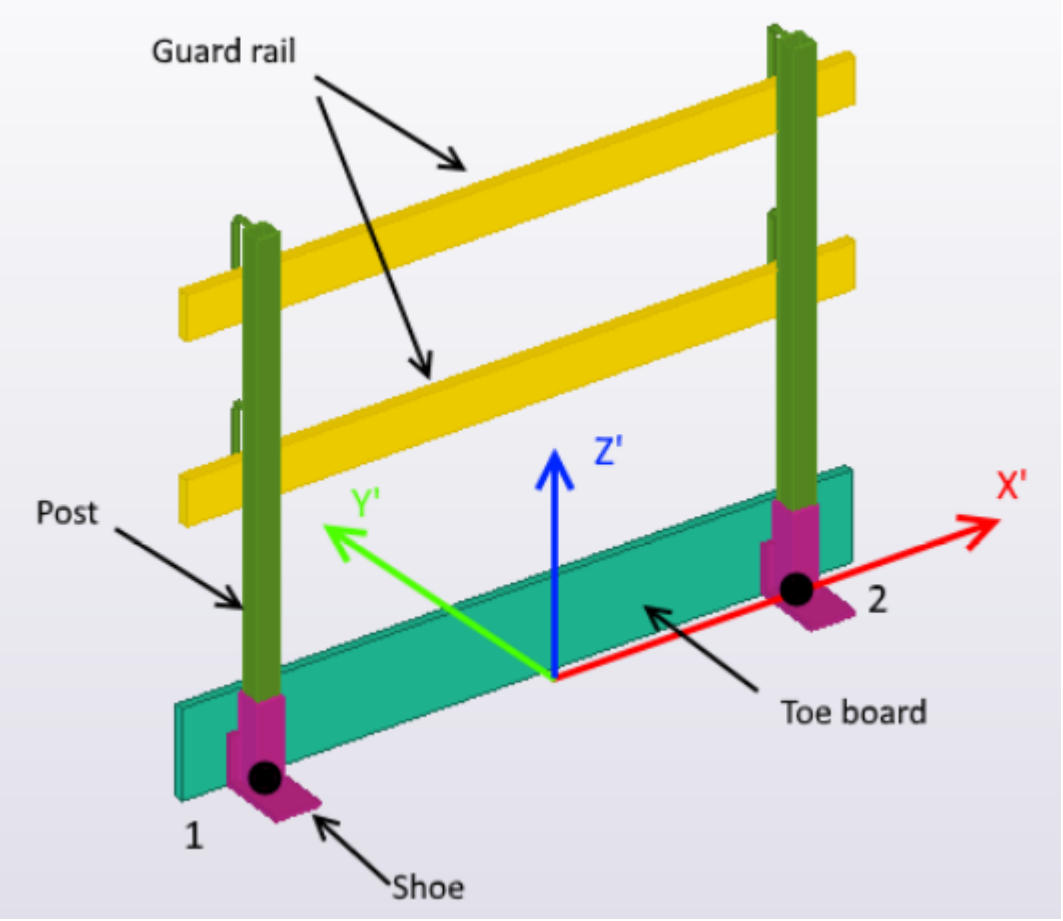
Row	Definition
	start point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
`AngleEndOffset`	This offset specifies the location of the second input point (of the custom part) in relation to the first input point in the local coordinate system. The offset is given as x, y and z values separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, zero offset (100 0 0) is used. `AngleEndOffset` specifies the location of the second input point of the custom part in relation to the first input point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, zero offset (100 0 0) is used.

Configure the railing custom parts for the railing placing tool

`xxxx.FormworkTools.Railings.csv`

For a railing, you can specify four custom parts. The post and the shoe parts are placed along the input line with the defined start distance and spacing. The toe board and one or more guard rail parts are placed between two posts. If the toe board and/or the guard rail part is long enough compared to the post spacing, they can span over three or more posts.

The position settings are given in the local coordinate system of the railing.



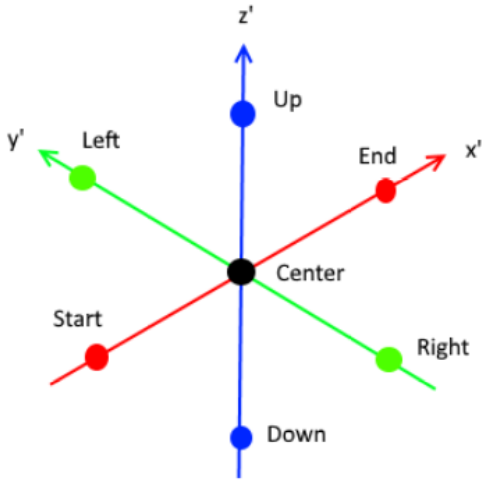
Row	Definition
`Supplier`	Supplier name, this is typically the same for all rows. The name is shown in the **Formwork supplier** list.
`Family`	Family name, which is shown in the **Product family** list.
`Name`	Railing type name, which is shown in the **Railing name** list. You can specify multiple railings with the same name when you have two or more alternatives with different lengths for the toe board and/or guard rail. When you have alternative lengths for the toe board and/or guard rail, the placing tool selects the part automatically based on the post spacing.

Row	Definition
`MasterInput`	The location of the first input point when placing the railing. The options are: • `PanelTop`: the first input is the slab panel component, and the railing is fitted on top of the panels. • `SlabTop`: the first input is the slab or pour top face. • `SlabEdge`: the first input is the slab or pour top face, and the railing location is the nearest edge. • `WallSide`: the first input is the wall side face. The railing is fitted on the wall side face. Note that if the `.csv` file does not contain any value, the default value `PanelTop` is used.
`MainOffset`	The common offset of all parts (post, shoe, and so on). The offset is a vector with components x, y, z given in the local coordinate system of the railing.
`PostSpacing`	Default spacing of the posts if no spacing is specified.
`PostSpacing`	Height of the post.
`ToeBoardOverlap`	Minimum overlap of toe boards.
`ToeBoardOverlapOffset`	Offset (x, y, z) of the toe boards when they are overlapping.
`GuardOverlap`	Minimum overlap of guard rails.
`GuardOverlapOffset`	Offset (x, y, z) of the guard rails when they are overlapping.
`GuardPositions`	The railing can include one or more rows of guard rails. This setting specifies the height (z) positions of the guard rail parts measured from the input line. Multiple values must be separated by a space or a colon, and enclosed in parentheses (450 650 850).

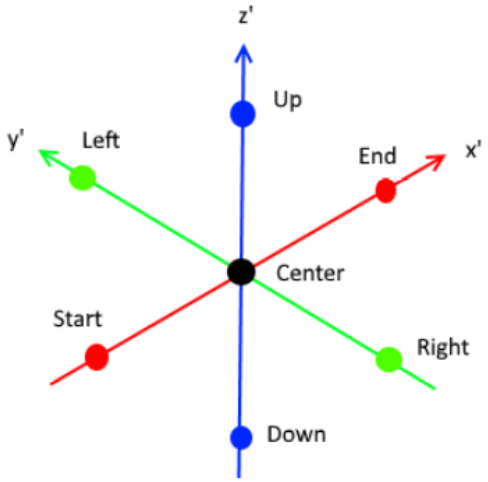
Row	Definition
PostName	Name of the custom part that will be inserted in the model.
PostAttributes	Property file saved in the component dialog box to be used when the custom part is inserted in the model.
PostPlanePosition	Use the following position values to define the custom part's position, offset and rotation in relation to the custom part start and end points. Use **Position in plane** to insert the custom part in the plane view. The options are **MIDDLE**, **LEFT**, and **RIGHT**.
PostPlaneOffset	Offset on plane. The default value is 0.
PostRotation	Use **Rotation** to insert the custom part in the plane view. The options are **FRONT**, **TOP**, **BACK**, and **BELOW**.
PostRotationOffset	Rotation offset in degrees. The default value is 0.
PostDepthPosition	Use **Position at depth** to insert the custom part in the plane view. The options are **MIDDLE**, **FRONT**, and **BEHIND**.
PostDepthOffset	Offset at depth. The default value is 0.
PostStartOffset	PostStartOffset is the offset of the actual start point from the generic start point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
PostEndOffset	PostEndOffset specifies the location of the second input point of the custom part in relation to the first input point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, zero offset (100 0 0) is used.

Row	Definition
ShoeName	Name of the custom part that will be inserted in the model.
ShoeAttributes	Property file saved in the component dialog box to be used when the custom part is inserted in the model.
ShoePlanePosition	Use the following position values to define the custom part's position, offset and rotation in relation to the custom part start and end points. Use **Position in plane** to insert the custom part in the plane view. The options are **MIDDLE**, **LEFT**, and **RIGHT**.
ShoePlaneOffset	Offset on plane. The default value is 0.
ShoeRotation	Use **Rotation** to insert the custom part in the plane view. The options are **FRONT**, **TOP**, **BACK**, and **BELOW**.
ShoeRotationOffset	Rotation offset in degrees. The default value is 0.
ShoeDepthPosition	Use **Position at depth** to insert the custom part in the plane view. The options are **MIDDLE**, **FRONT**, and **BEHIND**.
ShoeDepthOffset	Offset at depth. The default value is 0.
ShoeStartOffset	ShoeStartOffset is the offset of the actual start point from the generic start point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
ShoeEndOffset	ShoeEndOffset specifies the location of the second input point of the custom part in relation to the first input point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, zero offset (100 0 0) is used.

Row	Definition
ToeBoardName	Name of the custom part that will be inserted in the model.
ToeBoardAttributes	Property file saved in the component dialog box to be used when the custom part is inserted in the model.
ToeBoardLength	The effective length of the toe board part.
ToeBoardInputOrder	Set the final locations of the input points in relation to the default generic start and end points. The possible values are: • StartEnd: default if no other value is specified. • EndStart: reverses the default start/end points. • CenterStart: the first input point is at the center of the start/end point and the second point is at the start point. • CenterEnd: the first input point is at the center of the start/end point and the second point is at the end point. • CenterLeft: the first input point is at the center of the start/end point and the second point is at the left point. • CenterRight: the first input point is at the center of the start/end point and the second point is at the right point. • CenterUp: the first input point is at the center of start/end point and the second point is up from the center point. • CenterDown: the first input point is at the center of the start/end point and the second point is down from the center point. Note that if the .csv file does not contain any value, the default value StartEnd is used.

Row	Definition
	
`ToeBoardPlanePosition`	Use the following position values to define the custom part's position, offset and rotation in relation to the custom part start and end points. Use **Position in plane** to insert the custom part in the plane view. The options are **MIDDLE**, **LEFT**, and **RIGHT**.
`ToeBoardPlaneOffset`	Offset on plane. The default value is 0.
`ToeBoardRotation`	Use **Rotation** to insert the custom part in the plane view. The options are **FRONT**, **TOP**, **BACK**, and **BELOW**.
`ToeBoardRotationOffset`	Rotation offset in degrees. The default value is 0.
`ToeBoardDepthPosition`	Use **Position at depth** to insert the custom part in the plane view. The options are **MIDDLE**, **FRONT**, and **BEHIND**.
`ToeBoardDepthOffset`	Offset at depth. The default value is 0.
`ToeBoardStartOffset`	`ToeBoardStartOffset` is the offset of the actual start point from the generic start point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.

Row	Definition
ToeBoardEndOffset	ToeBoardEndOffset specifies the location of the second input point of the custom part in relation to the first input point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, zero offset (100 0 0) is used.
GuardRailName	Name of the custom part that will be inserted in the model.
GuardRailAttributes	Property file saved in the component dialog box to be used when the custom part is inserted into model.
GuardRailLength	The effective length of the guard rail part.
GuardRailInputOrder	Set the final locations of the input points in relation to the default generic start and end points. Note that the generic start and end points are located in the local z direction based on the setting. The possible values are: • StartEnd: default if no other value is specified. • EndStart: reverses the default start/end points. • CenterStart: the first input point is at the center of the start/end point and the second point is at the start point. • CenterEnd: the first input point is at the center of the start/end point and the second point is at the end point. • CenterLeft: the first input point is at the center of the start/end point and the second point is at the left point. • CenterRight: the first input point is at the center of the start/end

Row	Definition
	point and the second point is at the right point. • `CenterUp`: the first input point is at the center of start/end point and the second point is up from the center point. • `CenterDown`: the first input point is at the center of the start/end point and the second point is down from the center point. Note that if the `.csv` file does not contain any value, the default value `StartEnd` is used. 
`GuardRailPlanePosition`	Use the following position values to define the custom part's position, offset and rotation in relation to the custom part start and end points. Use **Position in plane** to insert the custom part in the plane view. The options are **MIDDLE**, **LEFT**, and **RIGHT**.
`GuardRailPlaneOffset`	Offset on plane. The default value is 0.
`GuardRailRotation`	Use **Rotation** to insert the custom part in the plane view. The options are **FRONT**, **TOP**, **BACK**, and **BELOW**.

Row	Definition
GuardRailRotationOffset	Rotation offset in degrees. The default value is 0.
GuardRailDepthPosition	Use **Position at depth** to insert the custom part in the plane view. The options are **MIDDLE**, **FRONT**, and **BEHIND**.
GuardRailDepthOffset	Offset at depth. The default value is 0.
GuardRailStartOffset	GuardRailStartOffset is the offset of the actual start point from the generic start point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, a zero offset (0 0 0) is used.
GuardRailEndOffset	GuardRailEndOffset specifies the location of the second input point of the custom part in relation to the first input point in the local coordinate system. Give the offset as x, y and z values that are separated by a space or a colon, and enclosed in parentheses (0 100 0). If no value is given, zero offset (100 0 0) is used.

Openings

This section introduces components that can be used in concrete openings.

Click the links below to find out more:

- Hole Generation (32) (page 3696)
- Polygon Hole Generation (33) (page 3698)

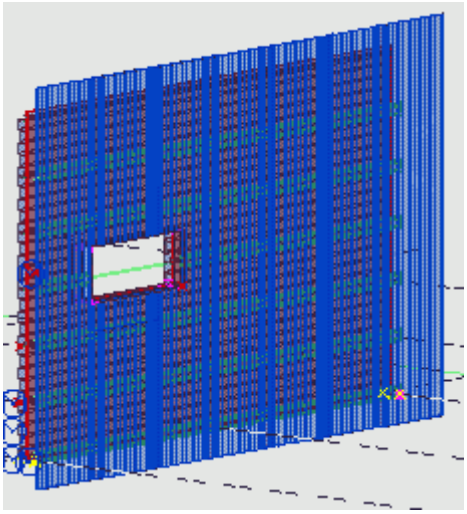
Hole Generation (32)

Hole Generation (32) creates a hole in the object, or splits the object in two if the hole cuts the whole object. Use this component, for example, in welded profiles, slabs, or panels generated with a component.

Objects created

- Cuts

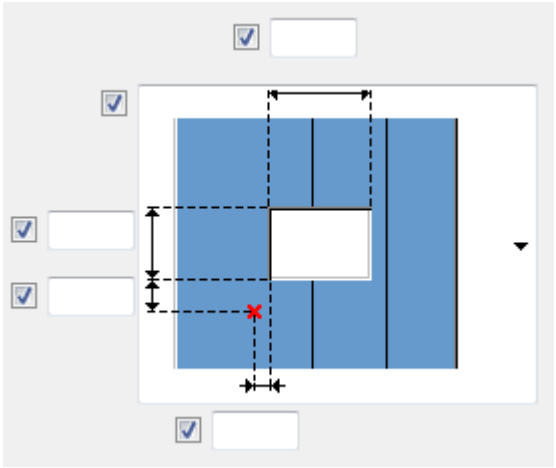
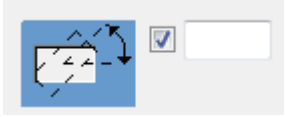
Use for

Situation	Description
	A hole is created when a point is picked.

Selection order

1. Select the part or object created by a component to be cut.
2. Click the middle mouse button.
3. Pick the position relative to the hole being generated.

Parameters tab
Use the **Parameters** tab to control the hole properties.

Option	Description
	Define the hole dimensions and the location of the picked point. The selected **Partcut** option affects which of these dimensions are available.
	To rotate the hole, define the rotation angle.

Option	Description
	Define the recess depth.
Partcut	Select the hole type. The options are: • **Rectangular** creates a rectangular hole using width and height. • **Circular** creates a round hole using height as a diameter. • **Profile** creates a cut using the selected profile. You can use this option to create a hole with an irregular shape.
Profile	If you selected **Profile** as the hole type, select a suitable profile from the **Profile Catalog**.
Coordination system	Select the coordination system. The options are: • **Use global xy-plane** creates the hole according to the global coordinate system. • **Use local** creates the hole according to the work plane. If the part is sloped or skewed, the **Use local** option is used automatically.
Cutpart name	Define a name for the cut part.
Rotated	Select whether the cut is rotated 90 degrees in the XY plane.

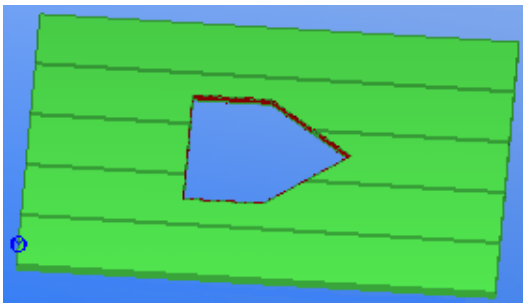
Polygon Hole Generation (33)

Polygon Hole Generation (33) creates a polygon-shaped hole in the object. Use this component, for example, in welded profiles, slabs, or panels generated with a component.

Objects created

- Cut

Use for

Situation	Description
	A polygon-shaped hole in concrete slabs, created by picking five points.

Selection order

To create a polygon-shaped hole:

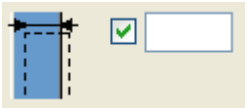
1. Select the part or object (component) to be cut.

   TIP You can cut other components, such as **Modeling of floor bay (66)** with this component.

2. Click the middle mouse button to end selecting.
3. Pick the corner points of the polygon-shaped hole.
4. Click the middle mouse button to end picking.

Parameters tab

Use the **Parameters** tab to control the cuts created.

Field	Description
	Cutting depth

Flooring

This section introduces components that can be used in concrete floors.

Click the links below to find out more:

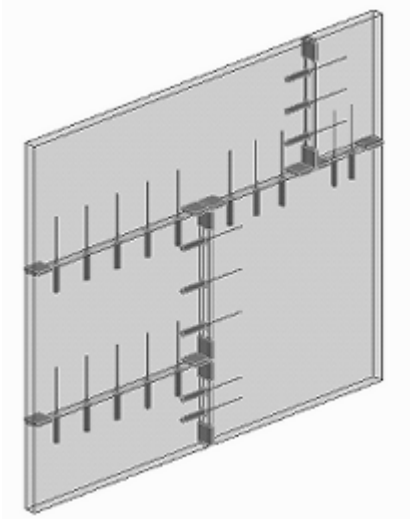
- Automatic seam recognition (30) (page 3700)
- Seam Applicator (page 3701)
- Modeling of floor bay (66) (page 3703)
- Sloping slab drainage (page 3711)
- Hollow Core Opening Tool (page 3715)

- Hollow Core Lifting Loops (page 3719)
- Floor layout (page 3722) (recommended for floor modeling)
- Floor Tool (page 3755)

Automatic seam recognition (30)

Automatic seam recognition (30) is used to add predefined custom seams between parallel parts, such as slabs or wall elements. This tool is useful when you add seams for warped floors.

Use for

Situation	More information
	Use to connect parts with predefined custom seams.

Before you start

Create parallel parts, for example, concrete slabs or walls.

Create custom seam.

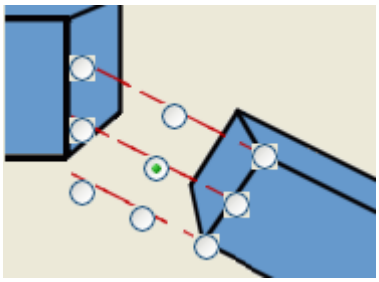
Selection order

1. Select the main part.
2. Select secondary parts. Click the middle mouse button to create the seams between the parts.

Parameters tab

Use the **Parameters** tab to define seam properties.

Property	Description
Seam name	Enter the seam name, or use the browse button (...) to locate the seam in the **Select component** dialog box.

Property	Description
Seam property file	Enter the name, or use the browse button (...) to locate the name of the seam attribute file (optional).
Seam direction	Select to change the direction.
Seam input part order	Select to reverse the main and secondary part. You may need to change the **Seam up direction** too.
Seam up direction	Define the direction of the seam.
Re-calculate seam points for	Use to re-calculate the seam points for all parts or only for warped parts. • **All parts**: use for fully or partially warped decks or floors. This option creates all seam points in the same way, no matter, if the slab is warped or not. • **Warped parts**: use for wall panels and other situations, where warping is not used, or the **All parts** option does not give desired result.
Position of re-calculated points	Control the location of the re-calculated seam points. The options in the list box are same as the circles in the following image. 

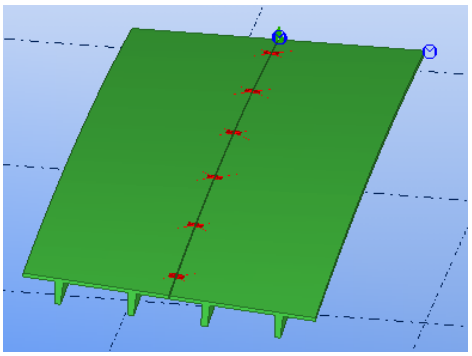
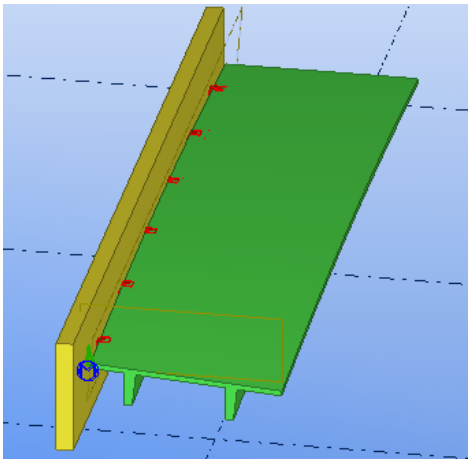
Seam Applicator

Seam Applicator creates a seam between two double tee slabs, or between a double tee slab and a wall or a spandrel. The double tees can also be warped and/or cambered. To be able to use **Seam Applicator** you need to have a custom seam. The custom seam can be created in the model, or it can be imported to the model.

Objects created

- Seams

Use for

Situation	Description
	Two double tee slabs with seams
	Double tee slab and wall with seams

Before you start

Create a custom seam.

NOTE When you define the custom seam in **Custom Component Wizard**, make sure that you select the **Allow multiple instances of connection between same parts** checkbox on the **Advanced** tab.

Selection order

1. Select the main part.
2. Select the secondary part.

   The seam is created automatically when the secondary part is selected.

   One of the selected parts must be a double tee slab and the other part can be a double tee slab, a wall, or a spandrel.

   NOTE If the seam cannot be created between the selected parts for some reason, a dummy beam is created instead. The beam indicates that the seam creation did not succeed.

Parameters tab
Use the **Parameters** tab to control the properties of the seam.

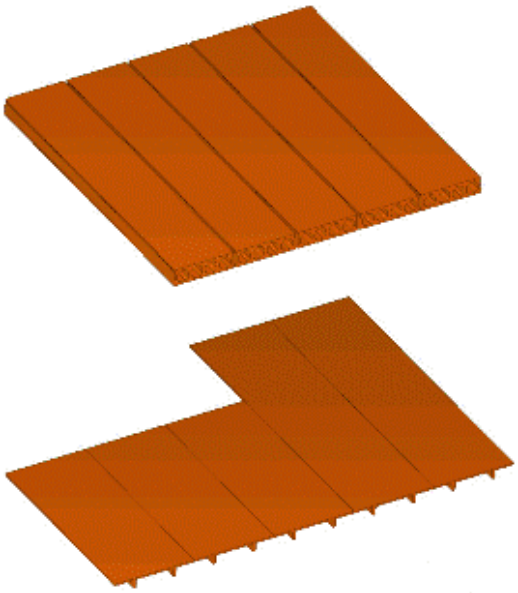
Option	Description
Seam name	Define the custom seam that connects the parts by selecting it from the **Applications & components** catalog.
Seam property file	Select the attribute file for the custom seam.
Seam direction	Seam direction in relation to the main and the secondary part.
Seam up direction	Seam rotation.
Offset start point	Seam start point offset from the edge of the double tee slab start point.
Offset end point	Seam end point offset from the edge of the double tee slab end point.
Number of copies	Number of seams.
Distance to first seam	Distance between the double tee slab edge and the first seam start point.
Spacing values	Space between the seams.
Copy at equal distances (Ignore spacing values)	Define whether seams are created at equal distances. If you select **Yes**, the values in **Spacing values** field are ignored.

Modeling of floor bay (66)
Modeling of floor bay creates an area of concrete slabs between two support lines that you define, for example, between two main frames. Support lines can be polylines or sloped.

Parts created

Concrete slabs.

Where to use

Situation	More information
	Slab area created between two support lines.

Before you start

Create two support lines with two or more points.

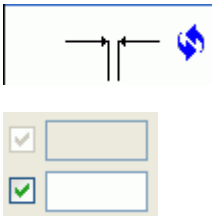
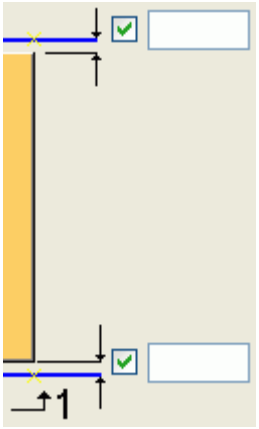
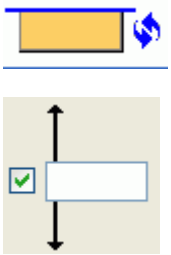
Picking order

1. Pick points to define the first support line. To finish, middle-click away from the last point picked.
2. Pick points to define the second support line. To finish, middle-click away from the last point picked.
3. Pick a point to indicate the origin of the slabs.
4. Pick a point to indicate the direction of the slabs.

Parameters tab

Use the **Parameters** tab to define the position of the slab.

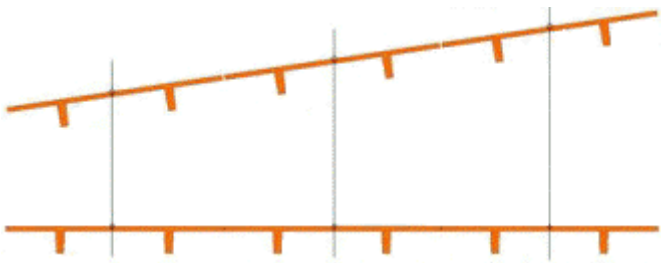
Property	Description
	The position of the slab relative to the origin line picked when creating the slab. The options are **center of profile** (default) or **center of seam**.

Property	Description
	Select the slab spacing and enter the appropriate dimension. The options are **seam** (default) or **center to center**. If you select the seam option, the bottom dimension is enabled. If you select the center-to-center otion, the upper dimension is enabled.
	Slab offsets from support lines 1 and 2.
	Vertical position of the slab. The options are **Top** (default), **Center** or **Bottom**. Enter an optional offset value.
Projection type	Defines how the slabs are spaced on a sloping floor bay.

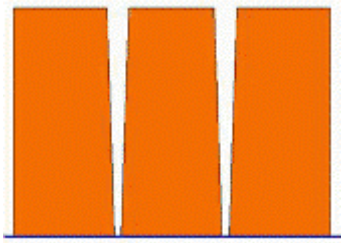
Projection type

Use the **Projection type** field to define how to handle slab spacing or seams on a sloping floor bay. The options are **Global** (default), **Support line 1**, **Support line 2**, or **Both support lines**.

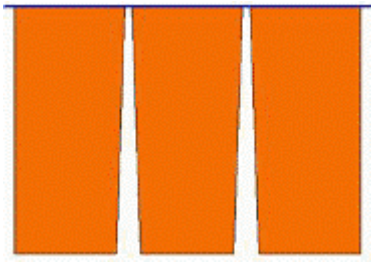
- **Global:** the centers of the slabs remain constant, seam spacing adjusts.



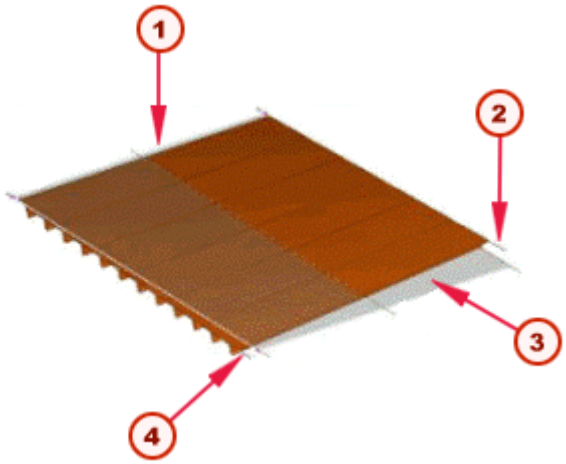
- **Support line 1:** Seam spacing on support line 1 does not adjust (effect exaggerated here):



- **Support line 2:** Seam spacing on support line 2 does not adjust (effect exaggerated here):



- **Both support lines:** Seam spacing is calculated at the average plane position between both support lines



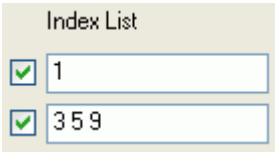
1. Along this line, center to center and seam width are exactly as entered in the dialog box.
2. Support line 2
3. Plane between support lines 1 and 2
4. Support line 1

Profiles tab
Use the **Profiles** tab to define the properties of the slab profiles.

Index list

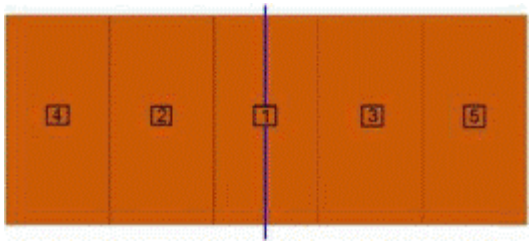
Use the **Index list** to specify different profiles, seam width, and various cutting options for specific slabs or seams.

Input the values individually, or in a list:

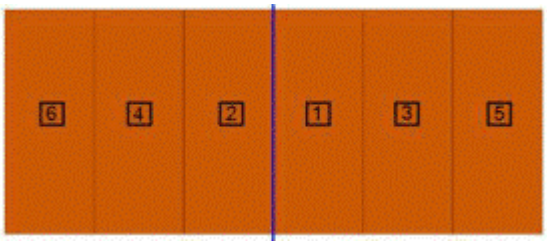


Examples for slabs

If the origin point is the center of slab:

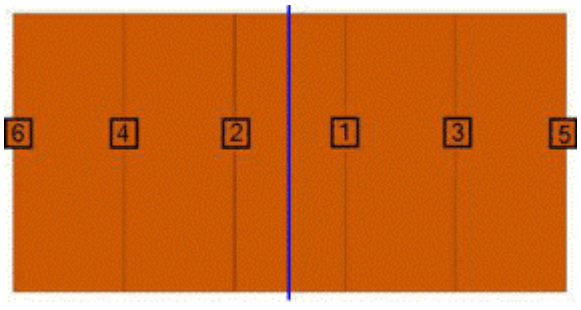


If the origin point is the center of seam:

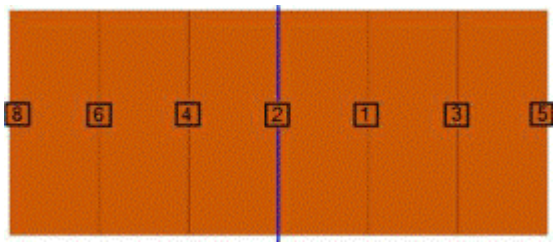


Examples for seams

If the origin point is the center of slab:



If the origin point is the center of seam:



Adjustment type

Use the **Adjustment type** to define the type of adjustment for the selected slab. The options are:

- **Default** - Use the default slab profile.
- **Profile** - Use the specific slab profile.
- **Cut left side** - Cut the left side of the profile. If no profile is defined, uses the default slab profile.
- **Cut right side** - Cut the right side of the profile. If no profile is defined, uses the default slab profile.
- **Cut both sides** - Cut both sides of the profile. If no profile is defined, uses the default slab profile.
- **Seam width** - Adjust the seam width.

Profile/Seam Width

Depending on which adjustment type option you choose, defines:

- The width of the default slab profile.
- The width of the specified profile.
- Seam width.

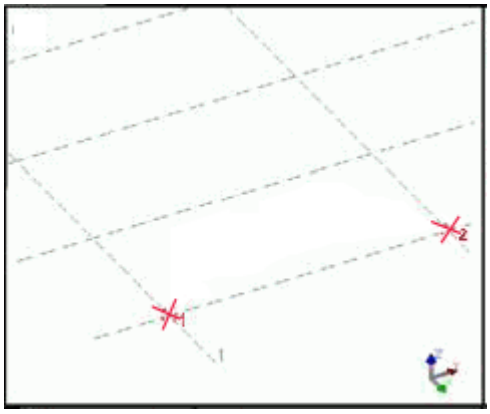
Examples

These examples show how to use the **Modeling of floor bay (66)** component.

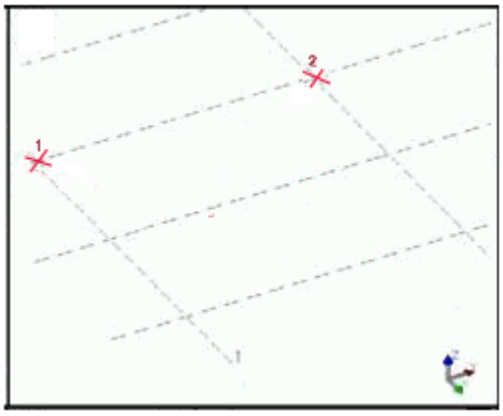
Example 1:

1. Pick 2 points (1, 2) to define support line 1.

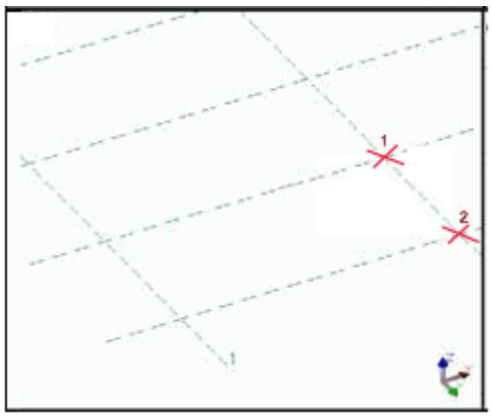
2. To finish, middle-click.



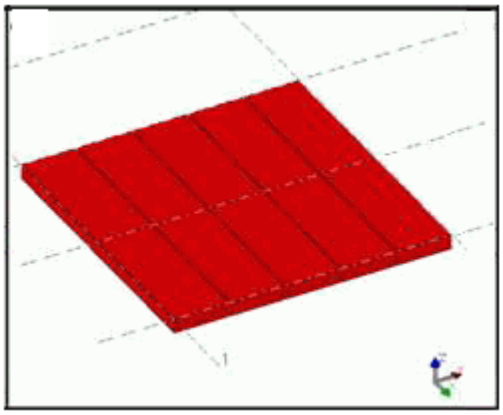
3. Pick 2 points to define support line 2.
4. To finish, middle-click.



5. Pick a point to indicate the origin of the slab.
6. Pick a point to indicate the direction of the slab.

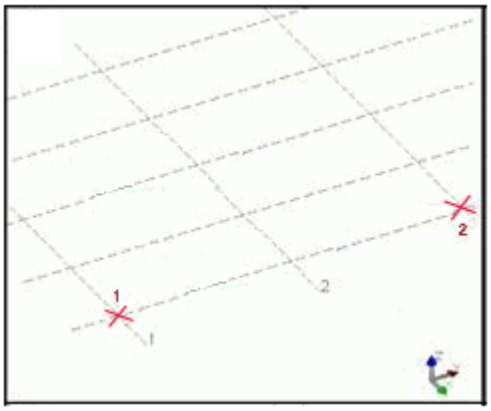


Tekla Structures creates the floor bay.

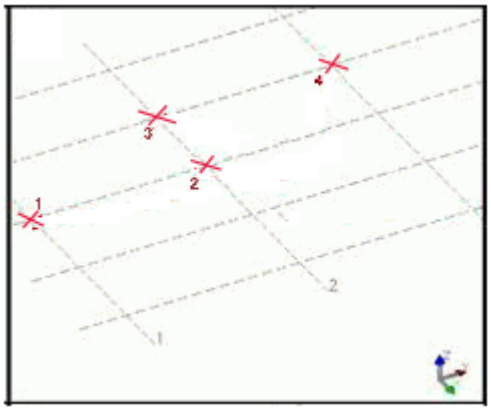


Example 2:

1. Pick 2 points to define support line 1.
2. To finish, middle-click.

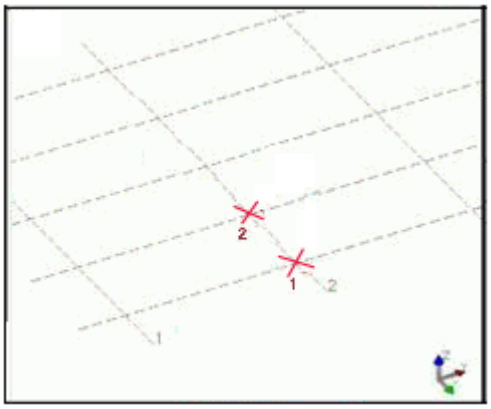


3. Pick 4 points to define support line 2.

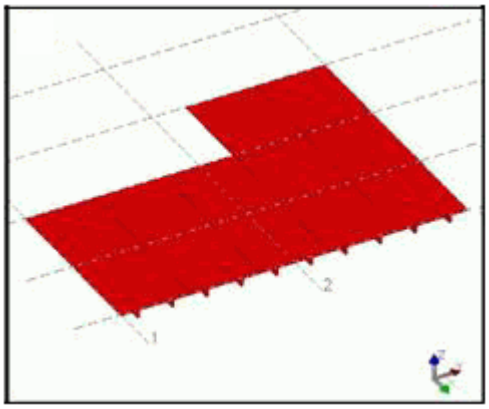


4. To finish, middle-click.
5. Pick a point to indicate the origin of the slab.

6. Pick a point to indicate the direction of the slab.



Tekla Structures creates the floor bay.



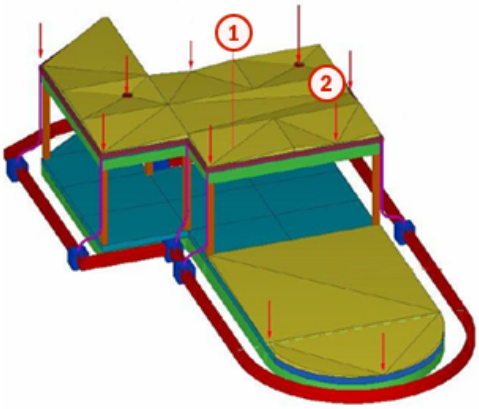
Sloping slab drainage

Sloping slab drainage creates a concrete slab topping and a drainage hole.

Objects created

- Drainage hole

Use for

Situation	Description
	Drainage holes with valley **1** and hip **2** created.

Limitations

Slab topping does not adapt to slab chamfers. You need to define the slab chamfers and topping chamfers separately.

Before you start

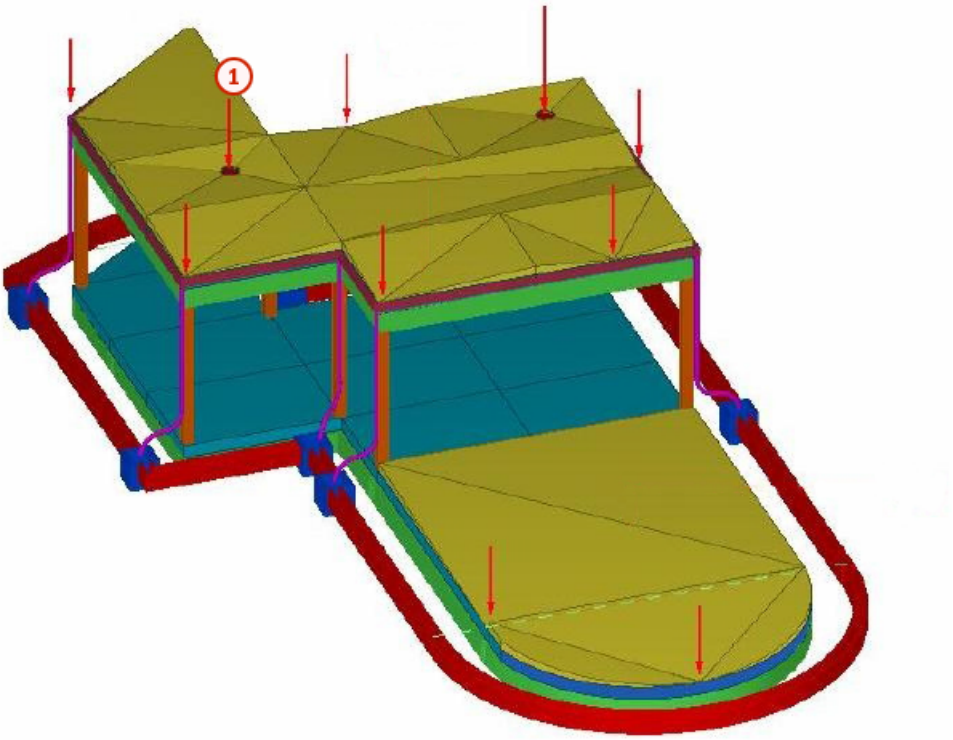
Create or split the slabs so that there is one drainage point in each slab.

Selection order

1. Select the main part (slab).
2. Pick a point to create the drainage hole.

   The drainage hole is created automatically when the point is picked.

Part identification key



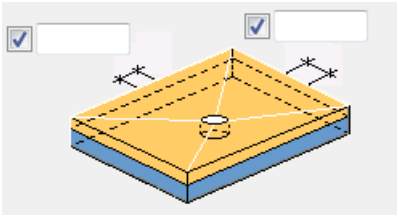
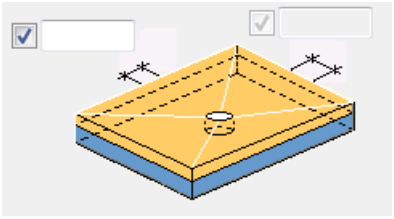
	Part
1	Drainage hole

Parameters tab
Use the **Parameters** tab to control the type, dimensions and properties of the drainage hole.

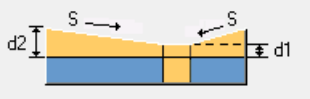
Drainage hole type

Option	Description
	A drainage hole is not created.
	A rectangular drainage hole is created.
	A round drainage hole is created.

Drainage hole dimensions

Option	Description
	Define the side dimensions of the rectangular drainage hole.
	Define the diameter of the round drainage hole.

Drainage hole properties



Option	Description
Define slab inclination by	Select how the slab is inclined, by a combination of slope (S) or slope percentage (S%), and topping thicknesses (d1, d2). The **Slope**, **d1**, and **d2** options are available accordingly.
Slope	Slope of the topping, for example, 0.01 (S) or 2 (S%).
d1	Thickness of the topping at the drainage point or at the hole.
d2	Thickness of the topping at the edge of the slab or at the hip.
Create cast unit	Select how the cast unit is created: • **Yes** Topping is part of the slab cast unit. • **No** Each part of the topping forms a cast unit.

Option	Description
	• **Topping** Topping is a separate cast unit.
Material	Define the material of the topping by selecting it from the material catalog.
Type	Select whether a valley or a hip is created. If you select **Hip**, you can only pick slab corners.

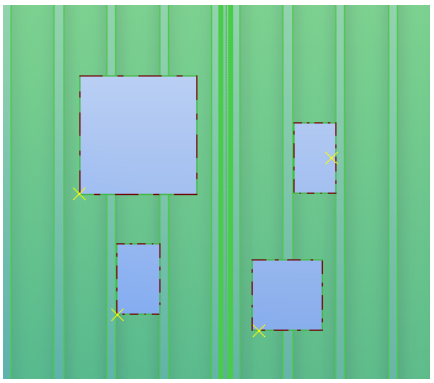
Hollow Core Opening Tool

Hollow Core Opening Tool creates holes and recesses to hollow core slabs. **Hollow Core Opening Tool** works with single hollow core slabs, or with floors that are created with the **Modeling of floor bay (66)** modeling tool.

Objects created

- Holes and recesses

Use for

Situation	Description
	Hollow core slab with small and large openings

Limitations

Hollow Core Opening Tool calculates the location for the openings based on the given values and the slab profile data. If needed, you can use an external configuration file in XML format to provide different rules for the openings and the slab profile data.

You can define small and large openings. Small openings are placed in the middle of the hollow cores so that no webs are broken. Large openings break webs.

Before you start

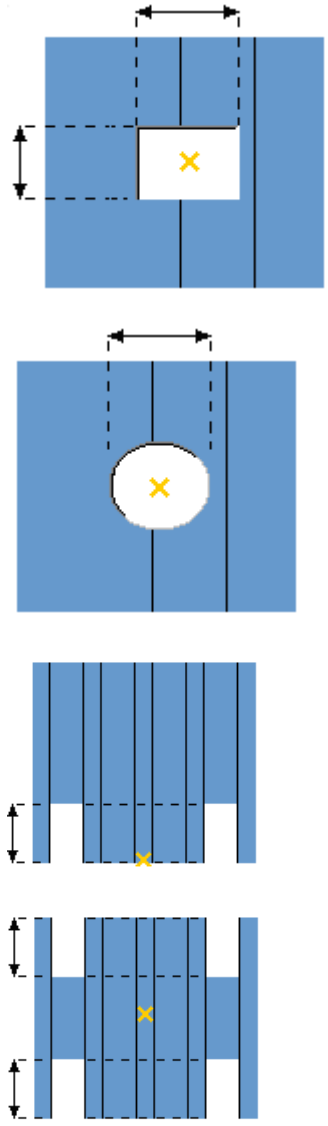
Create a hollow core slab.

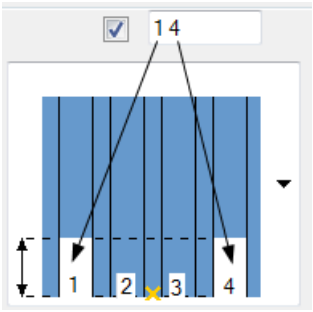
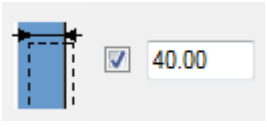
Selection order

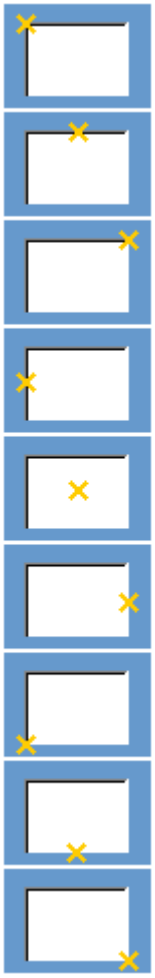
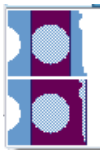
1. Select the main part (slab).
2. Pick a position for the opening.

Hollow Core Opening Tool properties

Use the **Hollow Core Opening Tool properties** to control the properties of the openings and recesses.

Option	Description
	Select the shape of the openings and recesses. • Square - in the middle of the slab • Round - in the middle of the slab • At one end of the slab • At both ends of the slab
Hollow positions	If you select to add openings or recesses at one end or at both ends of a slab, enter the positions of the hollows that are cut. The positions are numbered from left to right at the start of the part. The part is viewed from

Option	Description
	the top in the start-to-end direction. The openings and recesses are created in the same voids at the end of the part. 
Size	Define the size of the opening in longitudinal and cross direction, or the diameter for a round opening. Opening width defines whether the opening is small or large.
Opening Type	Select the type of the opening.
Place exactly by point	Select whether the opening is created exactly to the picked position.
Centralize in zone	Select how the center of an opening is centered • **By hollow**: The opening is centered to the hollow core. • **By web**: The opening is centered to the web.
Cut full width of hollow	Select whether the opening is cut based on the minimum width of hollows or the number of hollows. To use this option, center the opening **By hollow** in the **Centralize in zone** option.
Rules file (XML)	Select the external XML file where you have defined the opening and slab profiles.
Cutpart name	Define a name for the cut part.
	Define the recess depth.

Option	Description
	Define the alignment for the opening.
	Select whether the thick side part of the opening is removed.

Configuration file example
Use an external `.xml` file to define the openings and the slab profiles.

Example

An example `Zones.xml` file is located in `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\system`.

Zones

```
<ZoneRules>
        <!--Profile element has name and NumberOfSmall attributes which define number of small
zones in width direction. Profile name corresponds to TS profile name prefix that is P27, P32
etc. despite actual profile can be "P27(265x1200)" for example-->
        <Profile Name="P27" NumberOfSmall="5">
                <!--SmallZone element describes its parameters and must be the only in each
                profile. -->
                <SmallZone Start="77" Width="150" Interval="73" />
                <!--LargeZone element describes large zone (which breaks webs). There must be as
                many LargeZone elements as large zones supposed to be defined for a given
                profile. -->
                <LargeZone Start="325" Width="550" />
                <LargeZone Start="775" Width="425" />
                <LargeZone Start="0" Width="425" />
        </Profile>
        <Profile Name="P32" NumberOfSmall="4">
                <SmallZone Start="90" Width="180" Interval="95" />
                <LargeZone Start="375" Width="450" />
                <LargeZone Start="675" Width="525" />
                <LargeZone Start="0" Width="525" />
        </Profile>
</ZoneRules>
```

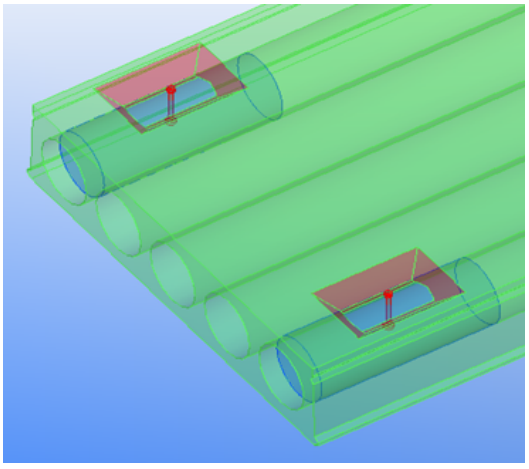
Hollow Core Lifting Loops

Hollow Core Lifting Loops creates lifters for hollow core slabs. **Hollow Core Lifting Loops** works with single hollow core slabs, or with floors that are created with **Modeling of floor bay (66)** modeling tool.

Objects created

- Lifters

Use for

Situation	Description
	Hollow core slab with recesses, hollow core fillings and lifting devices.

Limitations

You also need a lifting device. The lifting device can be created as a custom part, or it can be imported to the model. Alternatively, you can use an external configuration file in XML format to define the properties for the recess, the lifting device and the slab.

Before you start

Create a hollow core slab and a lifting device.

Selection order

1. Select the main part (slab).

   The recess and hollow core fillings are created.

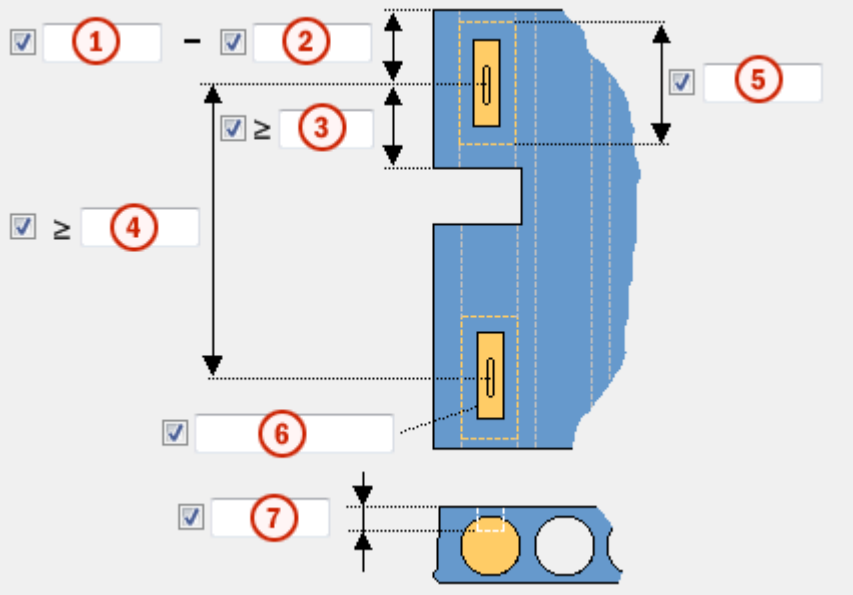
Basic settings tab

Use the **Basic settings** tab to control the properties of the lifters and hollow core fillings.

Configuration file

Option	Description
Configuration file	Select the external XML file where you have defined the recess, the lifting device, channel and grouting profiles, and the slab profile properties.
Override	Define whether the values on the **Basic settings** and **Lifting device settings** tabs override the properties defined in the external XML file.

Dimensions

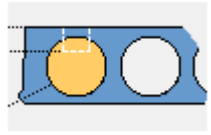
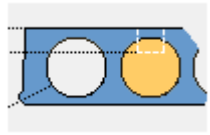
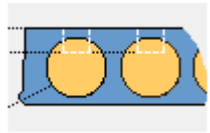


Option	Description
1	Define the minimum distance from the lifter insertion point to the end of the slab.
2	Define the maximum distance from the lifter insertion point to the end of the slab.

Option	Description
3	Define the minimum distance from the lifter insertion point to the edge of an opening.
4	Define the minimum distance between two lifter insertion points.
5	Define the length of the hollow core filling.
6	Define the profile of the recess.
7	Define the depth of the recess.

Channel numbers

Select the channel numbers:

Option	Description
	At the first channel Default
	At the second channel
	At all channels

Hollow core filling

Option	Description
Use automatic fill profile	Define whether the automatic hollow core filling profile is used.
Profile	Define the hollow core filling profile.
Prefix	Prefix for the part position number.
Start Number	Start number for the part position number.
Name	Define a name for the lifter. Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group the lifters. For example, you can display lifters of different classes in different colors.

Lifting device settings tab

Use the **Lifting device settings** tab to control the properties of the lifting device.

Move individual lifters in the x and y direction with direct modification.

Change the lifter type from the contextual toolbar.

Properties

Option	Description
Lifter component	Define the custom part that represents the actual lifting device.
Saved properties to use	Select which saved properties to use for the lifting device custom part.
Lifter max load	Define the maximum load for the lifting device.
Rotation	Select the rotation of the lifting device custom part around its axis.
Angle	Define the rotation angle of the lifting device custom part.
Up Direction	Select the lifting device up direction in relation to the local coordinate system.
Offset	Define the offset of the lifting device in relation to the local coordinate system in the corresponding directions.

Configuration file example

Use an external XML file to define the lifting device, the lifter, and the slab properties.

Example

Configuration file

```
<LiftingsConfiguration>
        </LiftingDevices>
                <Lifter Name="Lifter2" AttrFile="standard" MaxLoad="3000"
                RecessProfile="PRMD300*200-200*100"
                RecessDepth="60" UpDirection="+Z"
                Rotation="0" OffsetX="0" OffsetY="0" OffsetZ="0" />
                <Lifter Name="Lifter3" AttrFile="standard" MaxLoad="3000"
                RecessProfile="PRMD250*200-150*100" RecessDepth="50"
                UpDirection="+Z" Rotation="0" OffsetX="0" OffsetY="0" OffsetZ="0" />
        </LiftingDevices>
        <Profiles>
                <Profile Name="P27" MinEdgeDistance="200" MaxEdgeDistance="300"
                MinOpeningDistance="200" MinLiftersDistance="400"
                SideCoreDistance="0" CoreFillingLength="500"/>
                <Profile Name="P32" MinEdgeDistance="200" MaxEdgeDistance="300"
                MinOpeningDistance="200" MinLiftersDistance="400"
                SideCoreDistance="0" CoreFillingLength="500"/>
        </Profiles>
</LiftingsConfiguration>
```

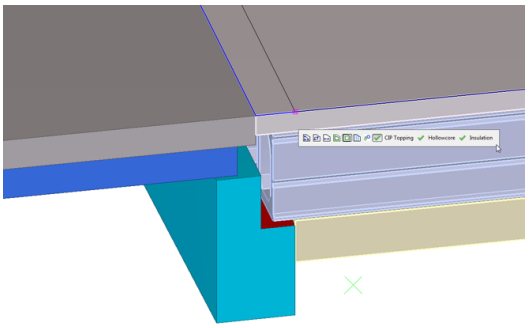
Floor layout

Floor Layout creates floors built of precast filigree, double-tee, and hollow core and solid slabs. The floors may contain several layers, such as precast slabs, insulation and a cast-in-place topping. **Floor Layout** is most useful for non-rectangular floors that contain one or more openings that split slabs into many pieces, and when slab widths are not constant on the whole floor area. You can modify the floors using direct modification.

Objects created

- Slabs
- Insulation
- Cast-in-place topping

Use for

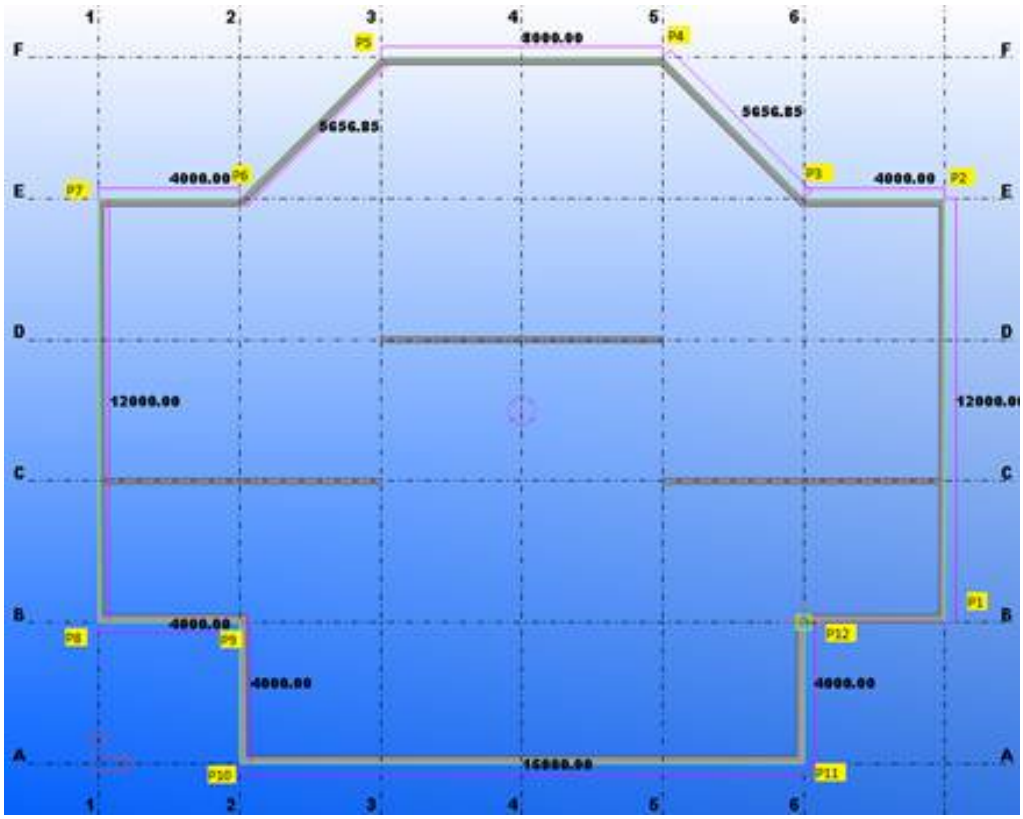
Situation	Description
	Create floor structures that have layers, for example, hollow core slabs at the bottom, insulation as the middle layer, and cast-in-place topping.

Selection order

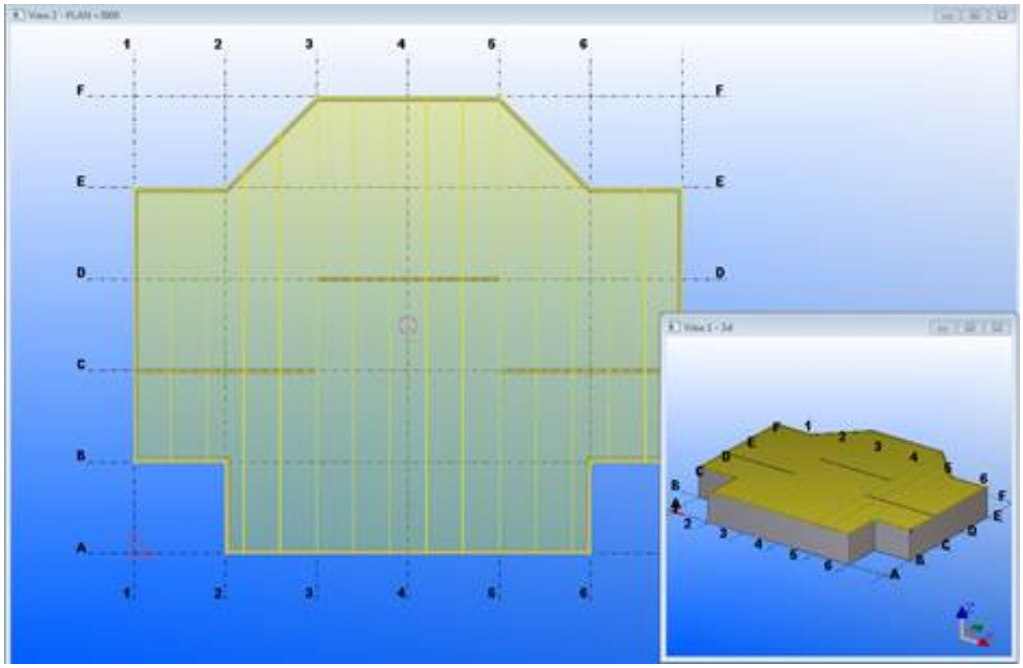
1. On the **Concrete** tab, click **Slab** --> **Floor layout** .
2. Pick the corner points of the floor.

   You can pick the points at grid lines or the reference points of supporting parts. You can change the offsets later.

   The default slab direction is from the first (P1) point to the second (P2) point. You can change this later if needed.



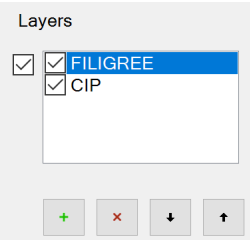
3. Click the middle mouse button to create the floor.



Layer tab

Use the **Layer** tab to control the layer type, profile, and how the layer is created.

Layer list

Layers	Description
	The layer list shows the current layers of the floor. The first layer in the list is the top layer and the last layer is the bottom layer. In the example image on the left, the `FILIGREE` layer is the top layer and the `CIP` layer is the bottom layer. Use the buttons to do the following: • Click to add a new layer. • Click to remove the selected layer. • Click to change the order of the layers by moving the selected layer up or down in the list. Use the check box in front of the layer name to control whether the properties of the selected layer are modified. Use the leftmost check box to control if the number of the layers and the order of the layers are modified when you modify the **Floor layout** component.

Layer properties

Select a layer in the layer list to define the layer properties.

Option	Description
Layer name	Enter a name for the layer. This is the layer name shown in the layer list. The name is also shown in the direct modification toolbar when you are modifying offsets . This name is not visible in the model so you can enter any suitable name.
Layer type	Select the layer type. The **Precast** layer is the primary layer and any insulation parts are added to precast units. The parts created for a **Cast in place** layer are not added to any other cast unit.
Create layer as	Select how the layer parts are created: as beam parts, custom parts, or a single part.
Layer component **Component attributes**	When the layer parts are created as custom parts, you can select a component and the component attributes.

Option	Description
Layer thickness or profile	When the layer parts are created as beam parts or as a single part, you can select the profile or thickness of the layer parts. With parametric profiles, you can replace the width of the profile with the text `[W]`. When you do this, the beam is never cut but the desired width is achieved by setting the profile name based on the slab width. For example, the profile could be `BL80*[W]`.
Rotation	Select the rotation of the layer parts.
Part name **Class** **Material** **Pour phase** **Part prefix**, **Start no** **Cast unit prefix**, **Start no**	Define the name, class, material, part prefix and start number of the layer parts. Define the pour phase for CIP layers.

General tab

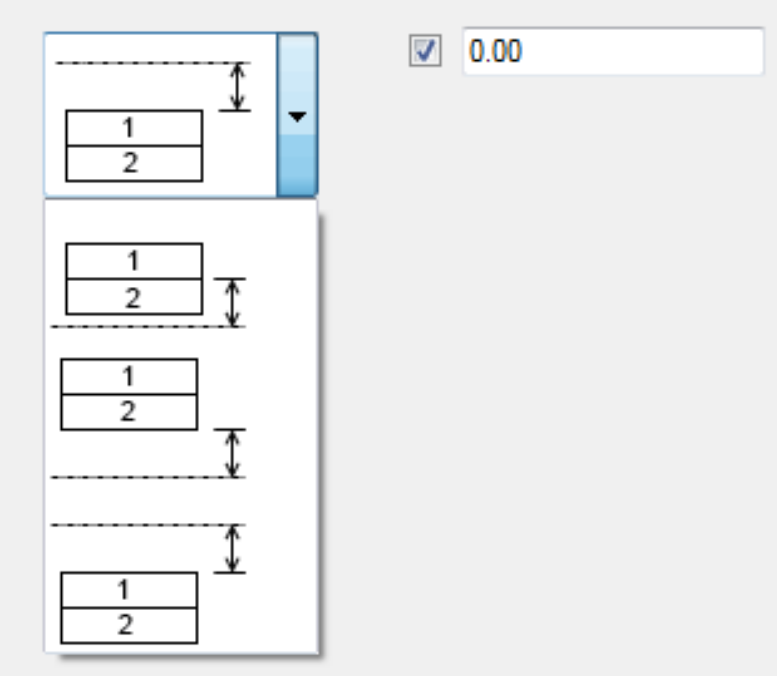
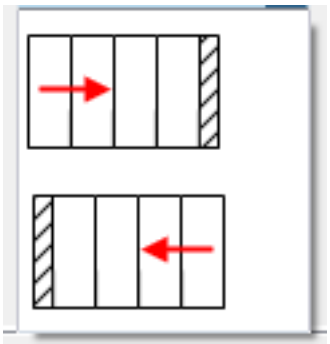
Use the **General** tab to control the default part and gap width, depth position of the layers, and the direction for lining up the slabs.

Select to create slabs and parts by width or number of them from the **Default part width** dropdown list. The options are:

Option	Description
Default part width	Enter the default width for the slabs in the primary layer. If you define the width of an individual slab in the slab part properties, the **Default part width** value is not used.
Max part width	Enter the maximum part width. **Floor layout** calculates the slab width by dividing the entire floor width by the maximum width. **Floor layout** then gets the number of slabs that are used to divide the entire floor width to get the optimal

Option	Description
	slab width taking the seam gap values into account in the calculation. Note that this setting is only recommended for profiles with parametric width (See also **Layer thickness or profile** in **Layer properties**), or for slab objects.
Number of parts	Enter the number of parts to be created. **Floor layout** calculates the slab width automatically by dividing the entire floor width by the number of parts. Note that this setting is only recommended for profiles with parametric width (See also **Layer thickness or profile** in **Layer properties**), or for slab objects.

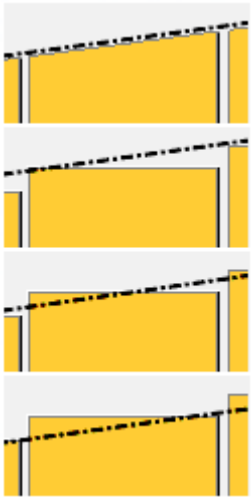
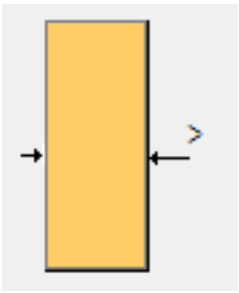
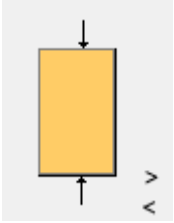
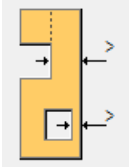
Option	Description
Default gap width	Enter the default seam gap width in the primary layer. If you define the width of an individual gap, the **Default gap width** value is not used.
Same gap for all	Set the gap width at first and last edges. **Same gap for all**creates gaps that are half of the default gap width. **No gap at first and last** does not create gaps.

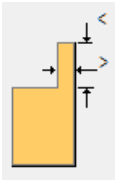
Option	Description
Depth position	Select how the layers are positioned in relation to the plane of input points. 
Line up direction	Select the direction for the slabs. By default, the width of the last slab may differ from the width of the other slabs. 

Advanced tab

Use the **Advanced** tab to control the minimum angles, and the width and length of the slabs.

Option	Description
Minimum end angle **Minimum start angle**	Controls how the slab ends are modeled. When the angle of the floor edge is smaller than the minimum angle, you can select a suitable option from the list to make the slab end straight.

Option	Description
	
Slab width and cut zone settings	Use this setting to select the `FloorLayout.ini` file that the component is using. The **Default** option is the file named as `FloorLayout.ini`. You can customize the allowed slab width, and the allowed locations of cuts and lengths in the `FloorLayout.ini` file. You can create multiple settings and save these settings in your own `xxxx.FloorLayout.ini` file, for example, `MyOtherSetting.floorlayout.ini`.
	Minimum slab width. Slabs narrower than the minimum width are not created.
	Minimum slab length. Slabs shorter than the minimum length are not created. Maximum slab length. Slabs longer than the maximum length are not created.
	Minimum required width of a single neck on either side of the slab when there is a large opening in the slab. If either of the necks on the sides of the opening is smaller than the defined neck width, the neck is cut off completely. In addition, the total width of the necks must be bigger than the minimum slab middle zone neck

Option	Description
	width, otherwise the slab is split into two separate slabs (the width of the actual opening is the same as the width of the slab).
	Minimum end nose width and length. If there are noses on both sides of the slab, these two end noses are checked separately and cut off if the width of the single nose gets smaller than the given minimum required nose width.
Align openings by allowed widths	Select **Yes** to re-align the opening/cut longitudinal edges to match the allowed width zones. The default value is **No**.
Create cut-outs	Select to create cut-outs for notches or openings. Selecting to create cut-outs does not affect whether the slab is split.
Extra narrowing	Define the gap increase between the slabs when the other slab is cut inside the layout. The default value is `0`.
Classes or names of parts to cut out	Creates automatic cut-outs for parts going through the floor layout. Enter the names or classes for parts that need cut-outs.

Customize allowed slab width and length

You can customize the allowed slab width, and the allowed locations of cuts and lengths in the `FloorLayout.ini` file. This file can be located under the model folder, or in any of the folders defined in the `XS_FIRM`, `XS_PROJECT`, or `XS_SYSTEM` advanced options.

If the file contains any values for a profile, these values are used and they override the minimum and maximum values entered in the **Floor layout** dialog box. In the **Floor layout** dialog box, you can define the slab width on the **General** tab and the slab length on the **Advanced** tab.

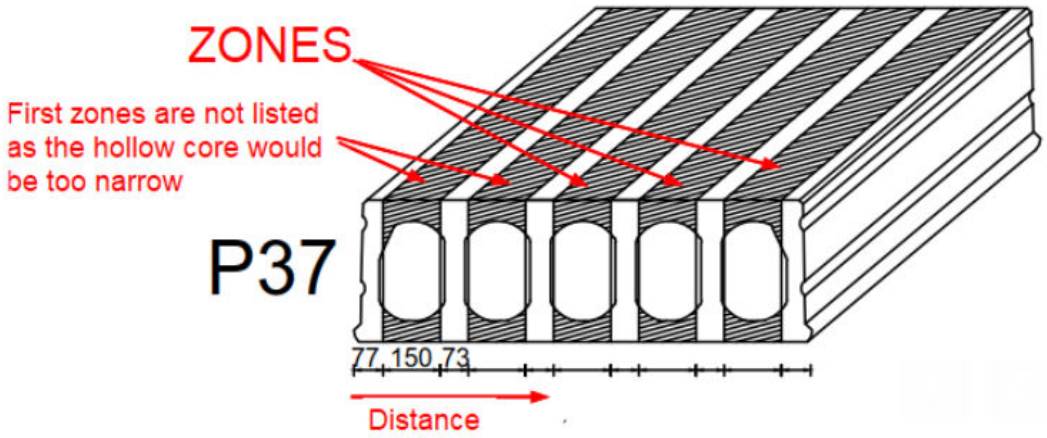
The images below show examples of the `FloorLayout.ini` file and allowed zones:

```
//
// configuration file for "Floor Layout" component
//
// Slab not listed below can have any width. Slab listed below can only
have width inside the given range.
// User input values will be rounded down to the nearest allowed value.
//
//                     Zone 1     Zone 2          Zone 3     Zone 4
Zone 5
// Slab widths    Profile    min. max.    min. max.     min. max.    min.
max.    min. max.
SLAB_WIDTH     P15(150X1200)   345-435     485-575     625-715
765-855      905-995      1045-1135
SLAB_WIDTH     P18(175X1200)   392-492     549-649        706-806
```

```
863-963      1020-1120
SLAB_WIDTH     P20(200X1200)   270-370     450-560     640-750
830-930      1020-1120
SLAB_WIDTH     P27(265X1200)   320-440     540-660     760-880
980-1110
SLAB_WIDTH     P32(320X1200)   380-560     660-830        930-1110
SLAB_WIDTH     P32R(320X1200)  380-560     660-830        930-1110
SLAB_WIDTH     P37(370X1200)   320-440     540-660        760-880
980-1120
SLAB_WIDTH     P40(400X1200)   400-530     670-800        950-1080
SLAB_WIDTH     P50(500X1200)   390-530     670-810        950-1080

//Allowed zones to place a cut
//                      Zone 1      Zone 2      Zone 3      Zone 4
Zone 5
// Cut zones    Profile     min. max.     min. max.     min. max.    min.
max.    min. max.
CUT_ZONES      P15(150X1200)   325-455     465-595     605-735
745-875    885-1015    1025-1155
CUT_ZONES      P18(175X1200)   372-512     529-669        686-826
843-983    1000-1140
CUT_ZONES      P20(200X1200)   250-390     430-580     620-770
810-950    1000-1140
CUT_ZONES      P27(265X1200)   300-460     520-680     740-900
960-1130
CUT_ZONES      P32(320X1200)   360-580     620-850        910-1130
CUT_ZONES      P32R(320X1200)  360-580     620-850        910-1130
CUT_ZONES      P37(370X1200)   300-460     520-680        740-900
960-1140
CUT_ZONES      P40(400X1200)   380-550     650-820        930-1100
CUT_ZONES      P50(500X1200)   370-550     650-830        930-1100
/
// Min/max lengths by slab profile & width min/max
//
//          Profile     Width min/max     Length min/max
SLAB_LENGTH      P15(150X1200)      0-1200      1000-8000
SLAB_LENGTH      P18(175X1200)      0-1200      1000-9000
SLAB_LENGTH      P27(265X1200)      0-1200      1000-12000
SLAB_LENGTH      P32(320X1200)      0-830       1000-13000
SLAB_LENGTH      P32(320X1200)      830-1200       1000-14000
SLAB_LENGTH      P32R(320X1200)        0-1200       1000-14500
SLAB_LENGTH      P37(370X1200)      0-660       1000-14000
SLAB_LENGTH      P37(370X1200)      660-1200       1000-15000
SLAB_LENGTH      P40(400X1200)      0-800       1000-16000
SLAB_LENGTH      P40(400X1200)      800-1200       1000-17000
SLAB_LENGTH      P50(500X1200)      0-810        1000-19000
SLAB_LENGTH      P50(500X1200)      810-1200     1000-20000
```

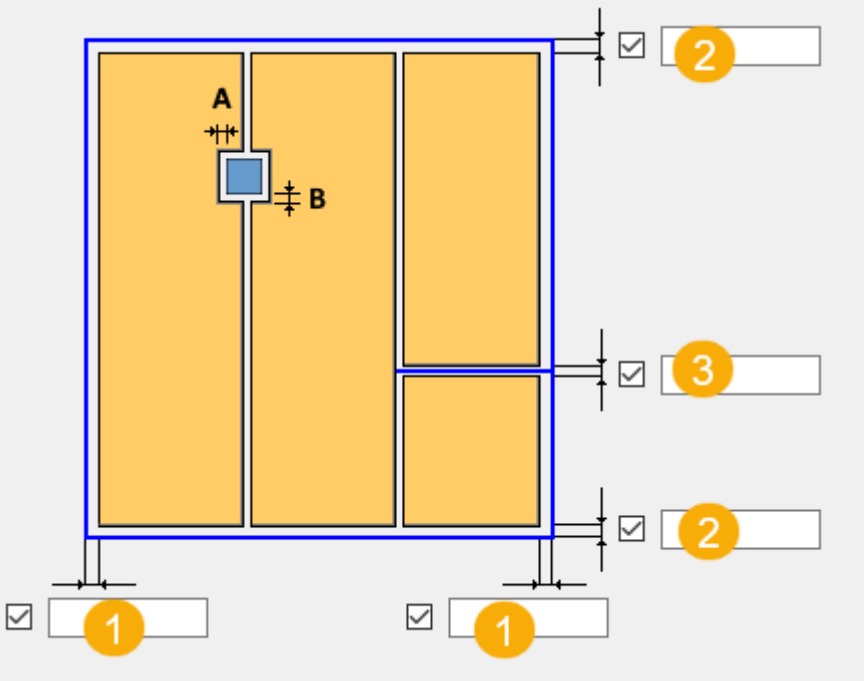
For example, in the following image, zone 1 is: `523-673 (77+150+73+150+73 - (+150))`.



You can create multiple settings and save these setting in your own `xxxx.FloorLayout.ini` file, for example, `MyOtherSetting.floorlayout.ini`.

Default offsets

Use the **Default offsets** tab to set the default offsets for each layer. These default offsets will be used whenever a specific value is not given for the edge.



	Description
1	Default offset for the side offset. Offset that is parallel to the bearing direction.

	Description
2	Default offset for the end offset. All other offsets that are not side offsets.
3	Default offset for the break line. The default value is `10`. If the break line value is `0`, the objects are not split.
A **B**	Define offsets for the automatically created cut-outs. You can define separate offsets for every layer.
Cutting around	Define how the automatic cut-outs are made in the selected layer: **By rectangle** or **By shape**. With **By shape**, the cut shape is the same as the shape of the part. For example, when the column is round, a round cut is created. You can also select not to create the cut. When you select **Do not create**, automatic cut-outs are not created in the layer.

Detailing tab

Use the **Detailing** tab to select the components that are applied to all slabs in the floor.

Option	Description
Component name	Select a component that is applied to the slabs in the floor.
Component attr file name	Select the settings that are used with the component you have selected. The list contains all the files that are available for the selected component.

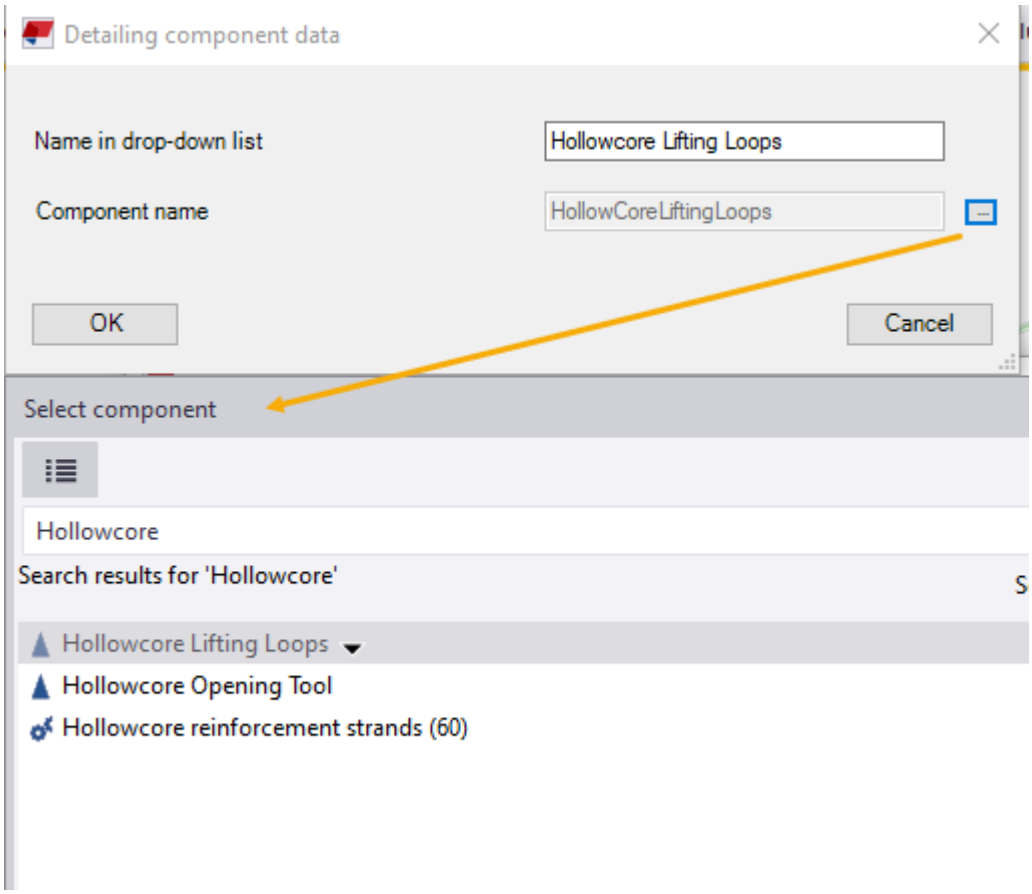
Adding a component to the component name list

The component name list is empty by default. You can add components to the list:

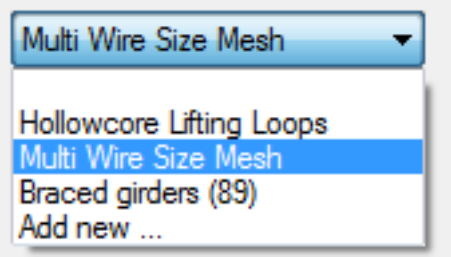
1. Select **Add new** from the list.

   The **Detailing component data** dialog box is displayed.
2. Enter a suitable name in the **Name in dropdown** box.
3. Click to open the **Applications & components** catalog.
4. Select a component and click **OK**.

   The component name or number is added to the **Component name** box.
5. Click **OK**.



When you have added the needed components, the names of the components are shown in the component name list as shown in the example image below. Select a component from the list:



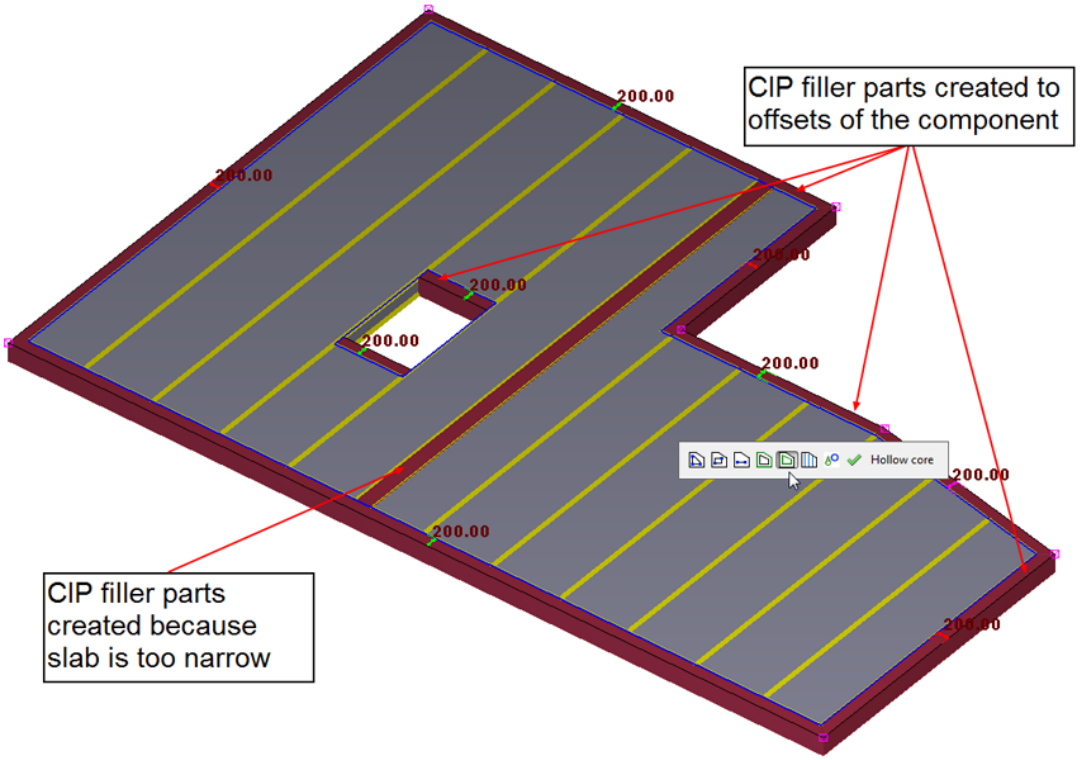
The list is stored in the `\attributes` folder under the model folder.

If you want to predefine the list to contain your favorite components in any new models, you can copy the `LayoutConfiguration.datx` file from the `\attributes` folder under the model folder to any of the folders defined in the `XS_FIRM`, `XS_PROJECT` or `XS_SYSTEM`.

CIP filler part tab

Use the **CIP filler part** tab to create CIP filler parts into the empty areas in floors created with **Floor layout**. The empty areas are spaces created by

offsets around the exterior and interior boundaries. There may also be empty areas when the slab is too narrow to be created with **Floor layout**.



CIP filler parts can also be created when:

- The allowed slab width is smaller than the needed width.
- The opening is larger than needed because of allowed width zones.
- The slab is split into two because of a large opening.
- The slab nose is not created because it is too narrow.

CIP filler part properties

Option	Description
Create filler parts	Select whether CIP filler parts are created.
Minimum width	Minimum width of the part. If the empty space is smaller than this width, the part is not created in this location.
Cast unit prefix	Cast unit prefix of the CIP filler part
Start no	Cast unit start number of the CIP filler part

Option	Description
Name **Class** **Material** **Finish** **Pour phase**	Define the name, class, material, finish and pour phase of the CIP filler parts.
Fill seam gaps	Select whether seam gaps are filled.

User defined tab
Use the **User defined** tab to set the user-defined attribute values for the slabs.

If you have defined the user-defined attributes in the part properties dialog box and want to use those values, leave the **User defined** tab empty.

The content of the tab may vary depending on your environment, see below how to customize the content.

Customizing the User defined tab

You can customize the content of the **User defined** tab using the `floorlayout.objects.inp` file. Note that you cannot create new user-defined attributes with the file. The user-defined attributes on the tab are existing user-defined attributes that have been defined in the `objects.inp` file. The `floorlayout.objects.inp` file has the same format as the `objects.inp` file.

The `floorlayout.objects.inp` file can be located in any of the folders set in the `XS_FIRM`, `XS_PROJECT` or `XS_SYSTEM` advanced options, or in the model folder.

Below is an example of the `floorlayout.objects.inp` file:

```
part(0,"Part")
{
     tab_page("Parameters")
     {
          attribute("comment", "j_comment", string, "%s", no, none, "0.0",
"0.0")
          {
               value("", 0)
          }
          attribute("xs_shorten", "j_xs_shorten", distance, "%d", yes,
none, "0.0", "0.0")
          {
               value("0.0", 0)
          }
          attribute("USER_FIELD_1", "j_user_field_1", string, "%s", no,
none, "0.0", "0.0")
          {
               value("", 0)
          }
          attribute("USER_FIELD_2", "j_user_field_2", string, "%s", no,
none, "0.0", "0.0")
          {
               value("", 0)
          }
          attribute("USER_FIELD_3", "j_user_field_3", string, "%s", no,
none, "0.0", "0.0")
          {
               value("", 0)
          }
          attribute("USER_FIELD_4", "j_user_field_4", string, "%s", no,
none, "0.0", "0.0")
          {
               value("", 0)
          }
     }
}
```

Modifying a floor

You can use direct modification to modify the floor. Before you start, ensure that the **Direct modification** switch is active. Select the floor to display the direct modification toolbar.

To modify the floor, select the appropriate command from the toolbar.

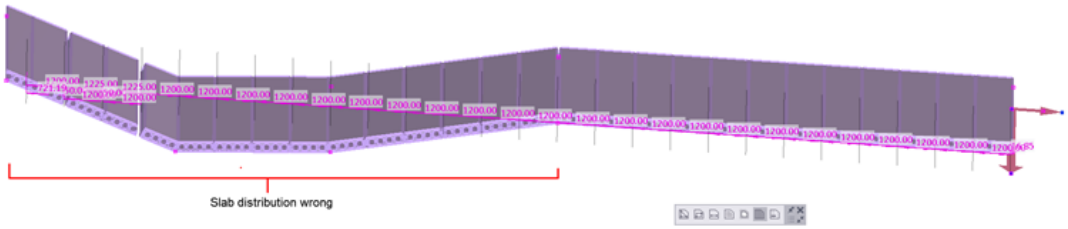
Button	Description
	Add a polygonal opening.
	Add a rectangular opening.
	Add a circular opening.
	Add a break line to split the slab into two slabs.

Button	Description
	Modify the openings or exterior boundary of the floor. This command is active by default. Make the floor warped by offsetting one or more vertices of the exterior boundary polygon from the plane.
	Modify the offsets at floor boundaries.
	Modify the layout of the floor: • Modify the width or profile of any of the slabs in the floor. • Modify the side of the cut when the slab is cut to a smaller width. • Modify the gaps between the slabs. • Modify the direction of the slabs. • Modify the order of slabs when the slabs do not have the same width or profiles. You can click the button to reset all seam gaps and slab distribution to the initial default values that were used when the floor layout was created.
	Add or modify the detailing components that are only applied to certain slabs in the floor layout. The detailing components are listed on the **Detailing** tab. You can apply the same detailing component settings to all slabs in the floor layout using the floor layout attributes.
	Click the refresh command to rerun the component. For example, if you have new parts that should cut holes around them, the refresh command ensures that the cuts are shown correctly.

Known limitations

Floor layout is not designed to work with floors where the layout is not in one plane. Egdes can be offset from the plane only to warp the floor, not to make floors that have several planes.

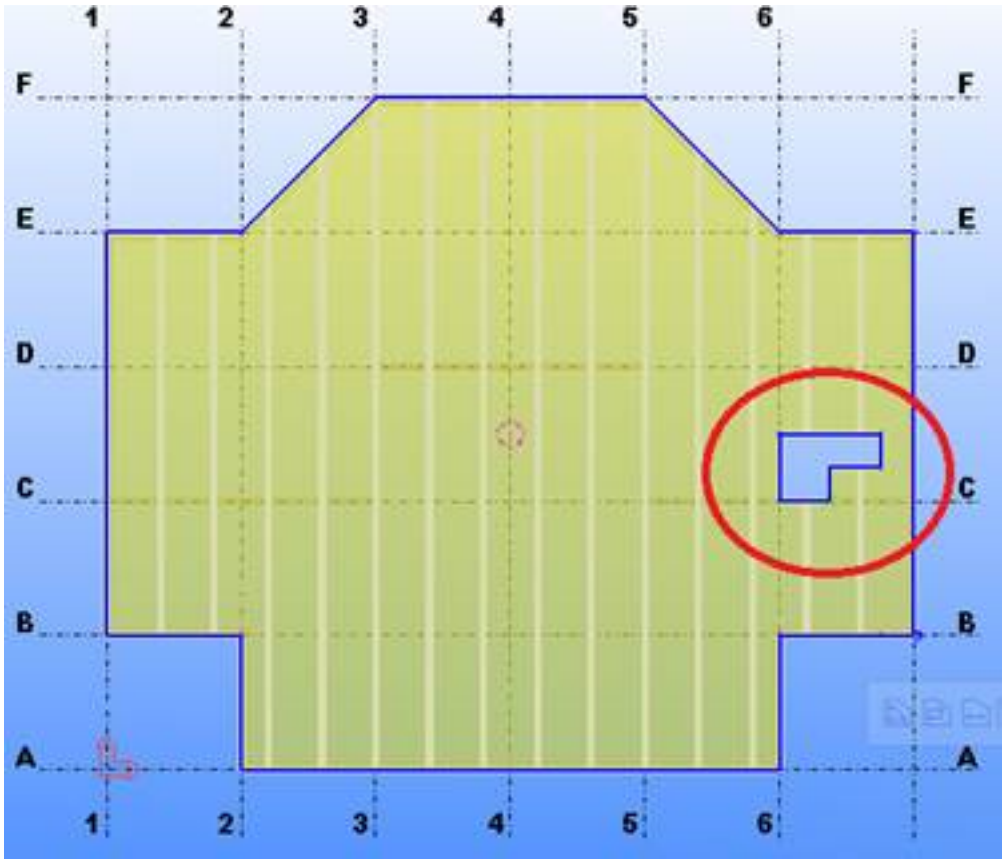
The image below shows an example where **Floor layout** has been used to create several planes. If precast slabs are used in this way, the outcome is not correct. Use multiple **Floor layout** components to create precast floors with several planes.



Add a polygonal opening

1. Click .
2. Pick the corner points of the opening.

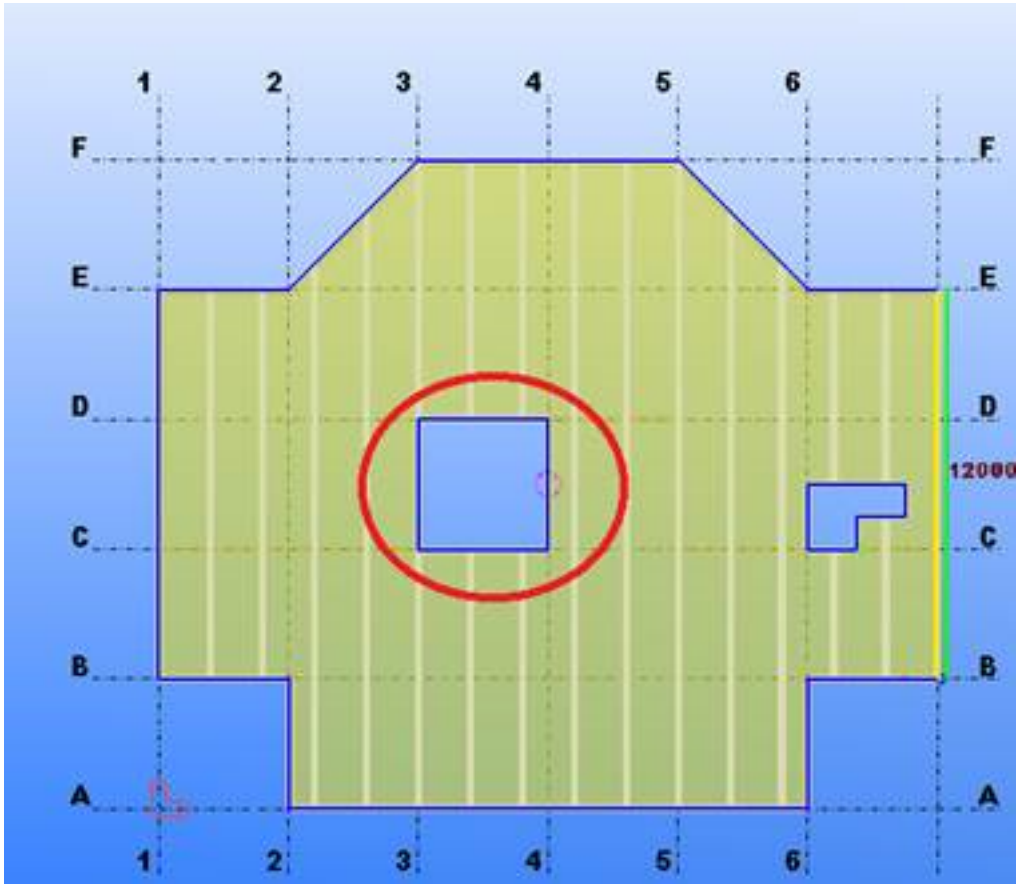
   You can pick the points at grid lines or you can pick the reference points of supporting parts. You can change the offsets later.



Add a rectangular opening

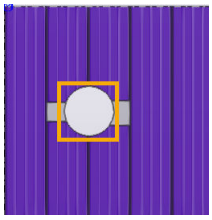
1. Click .
2. Pick the corner points of the opening.

   You can pick the points at grid lines or you can pick the reference points of supporting parts. You can change the offsets later.



Add a circular opening

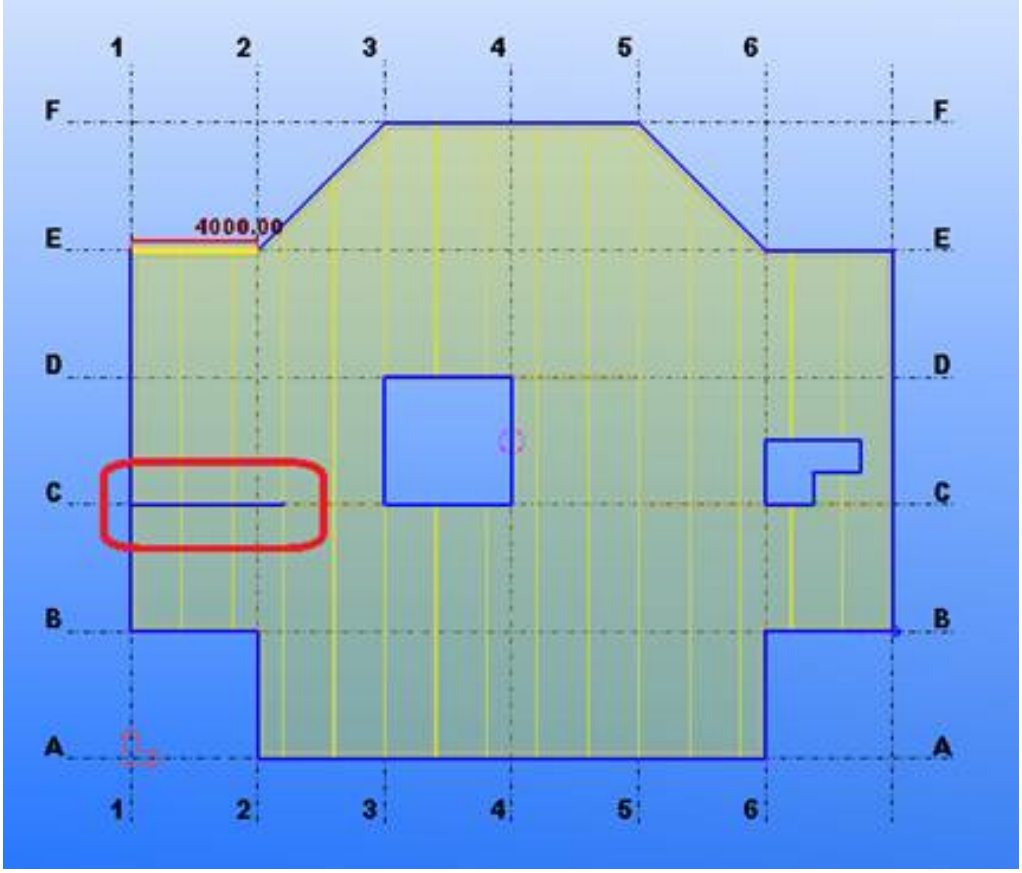
1. Click .
2. Pick the center point of the opening.
3. Pick the radius of the opening.



Add a break line to split the slab into two slabs

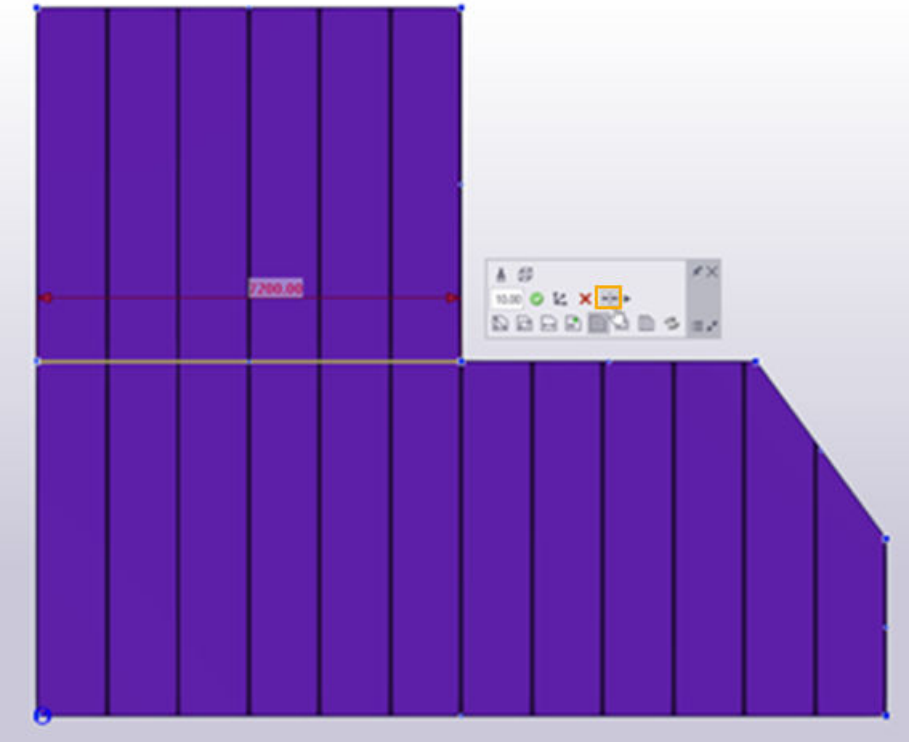
1. Click .

2. Pick the end points of the break line.



Split the floor layout to two separate floor layouts

1. Select either a seam or a break line.
2. Click on the contextual toolbar.



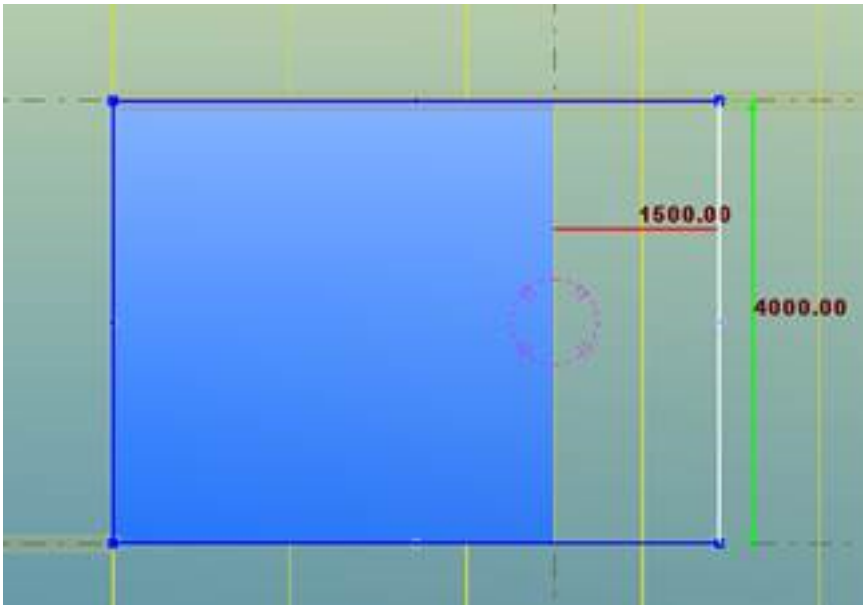
After the splitting, you can modify both layouts independently. Note that splitting cannot be reverted, and you cannot combine the two layouts again.

Modify the openings and exterior boundary of the floor

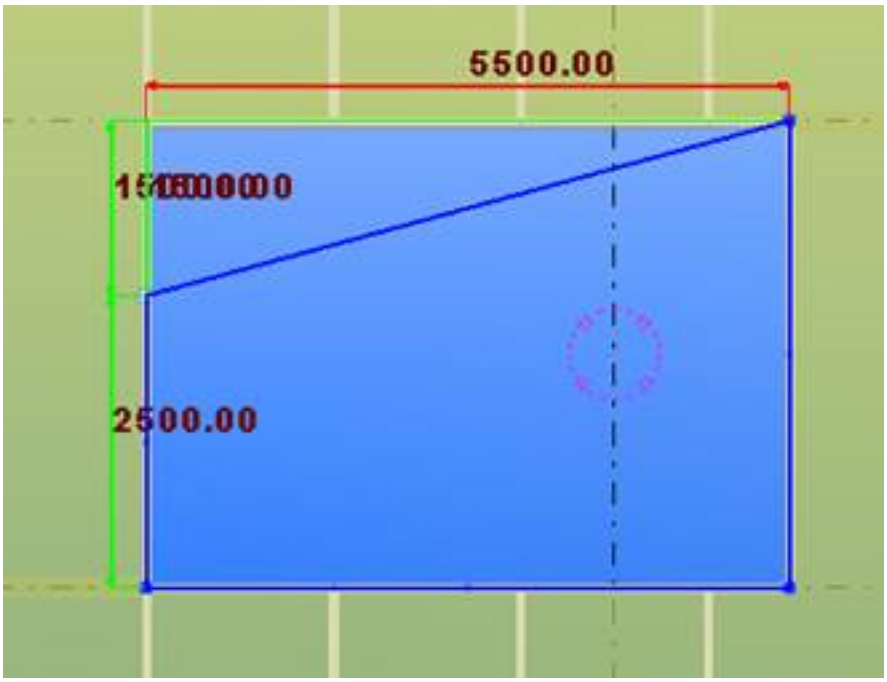
1. The button is active by default.

   The handles are shown for the exterior boundary and for all openings in the floor.

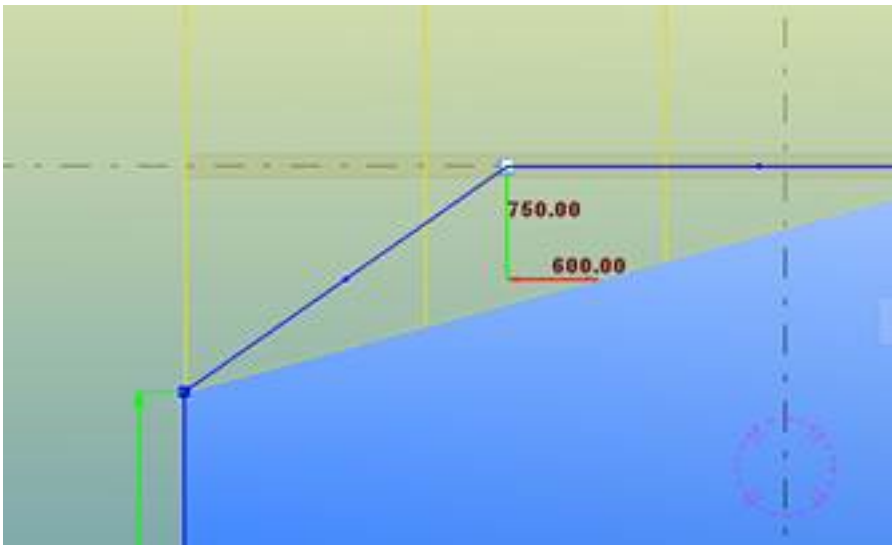
2. Do any of the following:
   - To move an edge, drag any of the edge lines. Hold down the left mouse button when dragging.



   - To move a corner point, drag the handle in the corner.

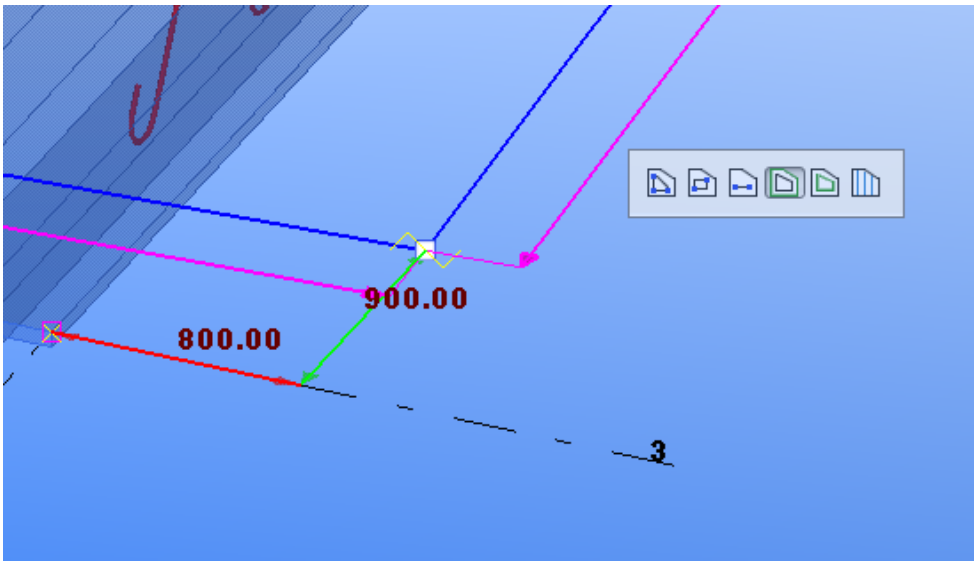


- To add new points to the edge, drag the handle in the middle of the edge.

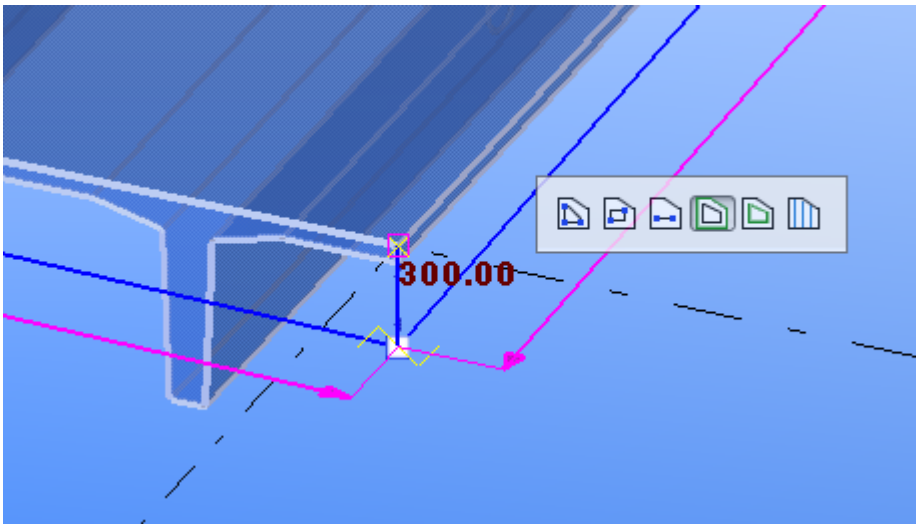


- To delete any of the points, select the handle in the corner and press the **Delete** key.
- To make the floor warped, offset the vertices of the exterior boundary so that all vertices are not in the same plane.

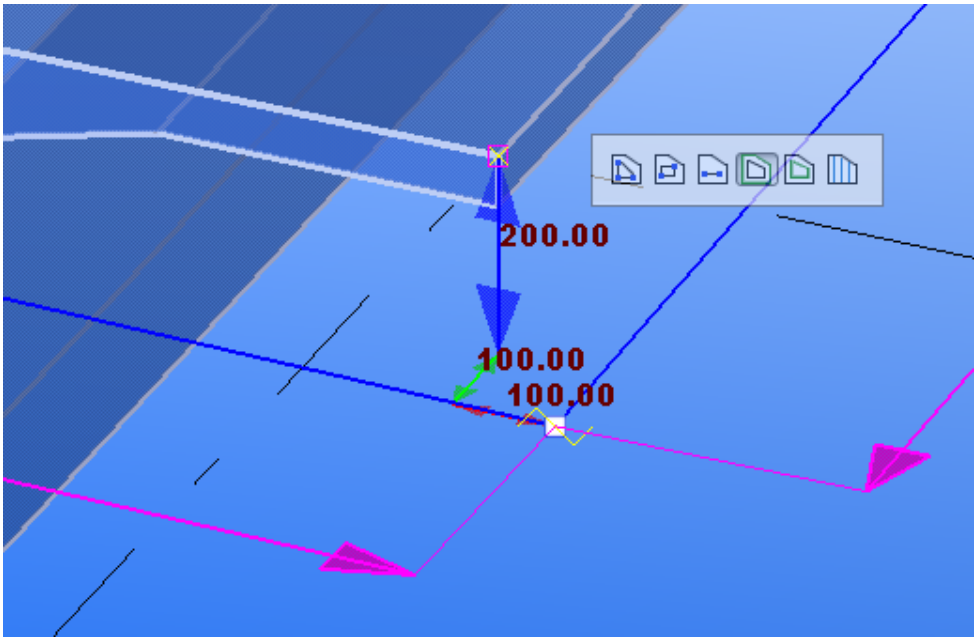
  Drag a handle normally to move the vertices in the original floor plane.



  Hold down the **Alt** key while dragging to move the vertex perpendicular to the floor plane.



Hold down the **Alt** and **Shift** keys to move the vertex both in the plane and off from the plane by snapping to any point in the model.

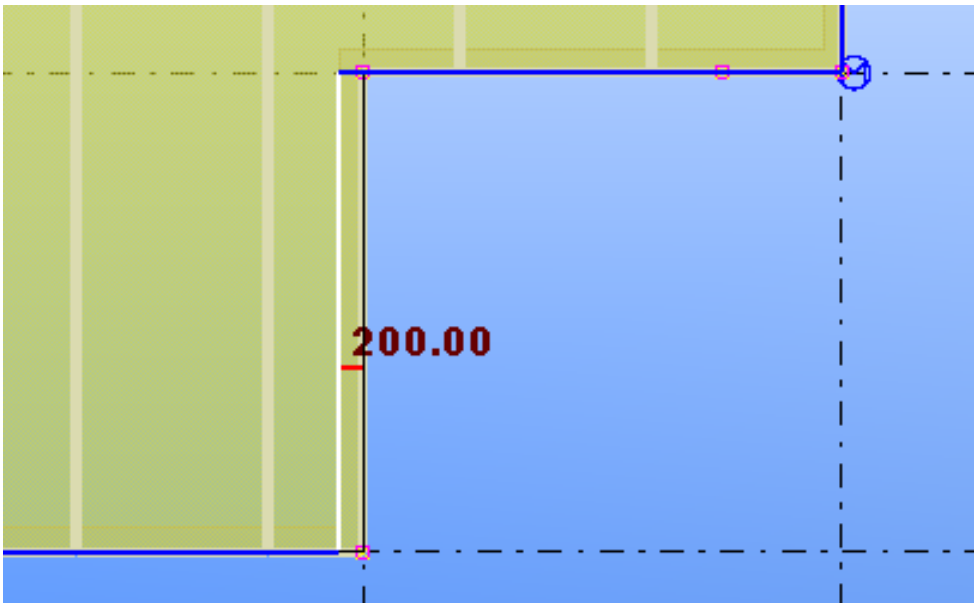


Modify the offset at floor boundaries

1. Click .

   The handles for modifying the offset are shown. If there are offsets, the offset dimensions are also shown.

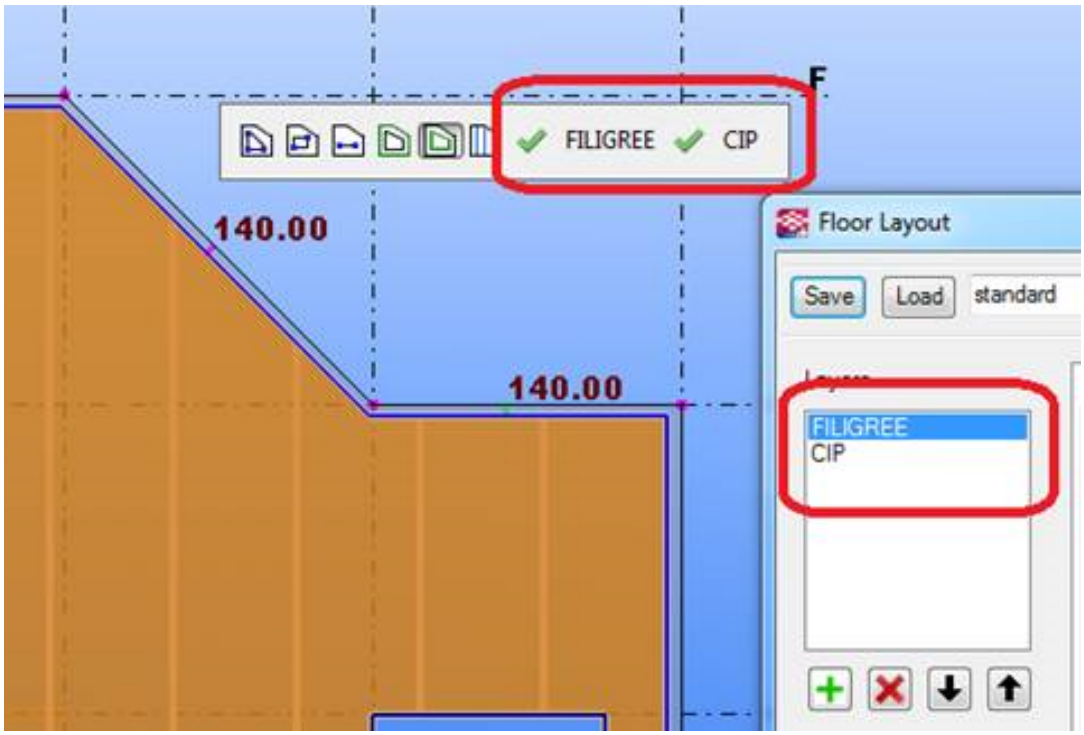
2. To adjust the offset, do one of the following:
   - Drag the edge offset lines.



   - Select one or more offset lines by holding down the **Shift** key.

     When you have the offset lines selected, right-click on any of the selected lines. Enter the offset value to the toolbar that is displayed and click .

Note that when you use the offset command, the direct modification toolbar shows the floor layout layers. Each layer may have different offsets. You can select which layers to modify. The simplest way is to select all layers and have the offsets equal in each layer.

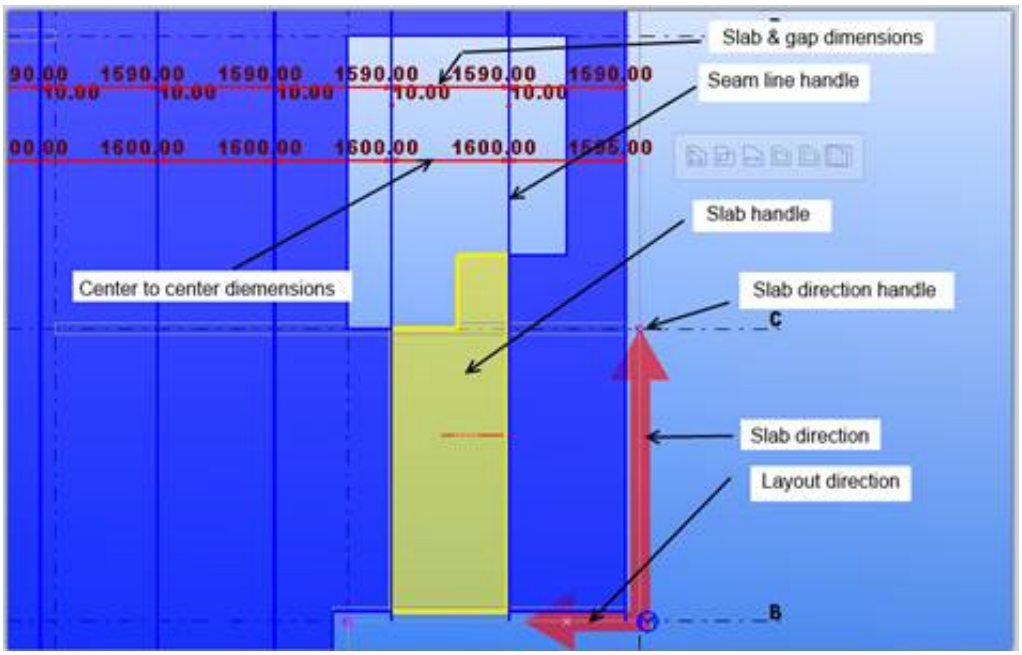

F
FILIGREE
CIP
140.00
140.00
Floor Layout
Save
Load
standard
FILIGREE
CIP

F
140.00
140.00
140.00
E
140.00
140.00
FILIGREE
CIP
D
50.00
50.00
200.00
50.00
50.00
C
140.00
140.00
B
200.00
200.00
140.00
A

Modify the layout of the floor

1. Click .

   The following direct modification elements are shown:



2. To modify the layout of the floor, do any of the following:
   - To modify slab width, do one of the following:
     - To move all seams, select one seam and start dragging.
     - To add a new seam, select a seam, hold down the **Ctrl** key and start dragging. When you stop dragging and release the mouse key, a new seam is added to that location.
     - To make one slab smaller, select a seam, hold down the **Alt** key and start dragging. The slab on the left or right side is made smaller and all slabs on the other side will move with the seam.
   - To modify the profile, width, and cut side of a slab, right-click the slab handle.

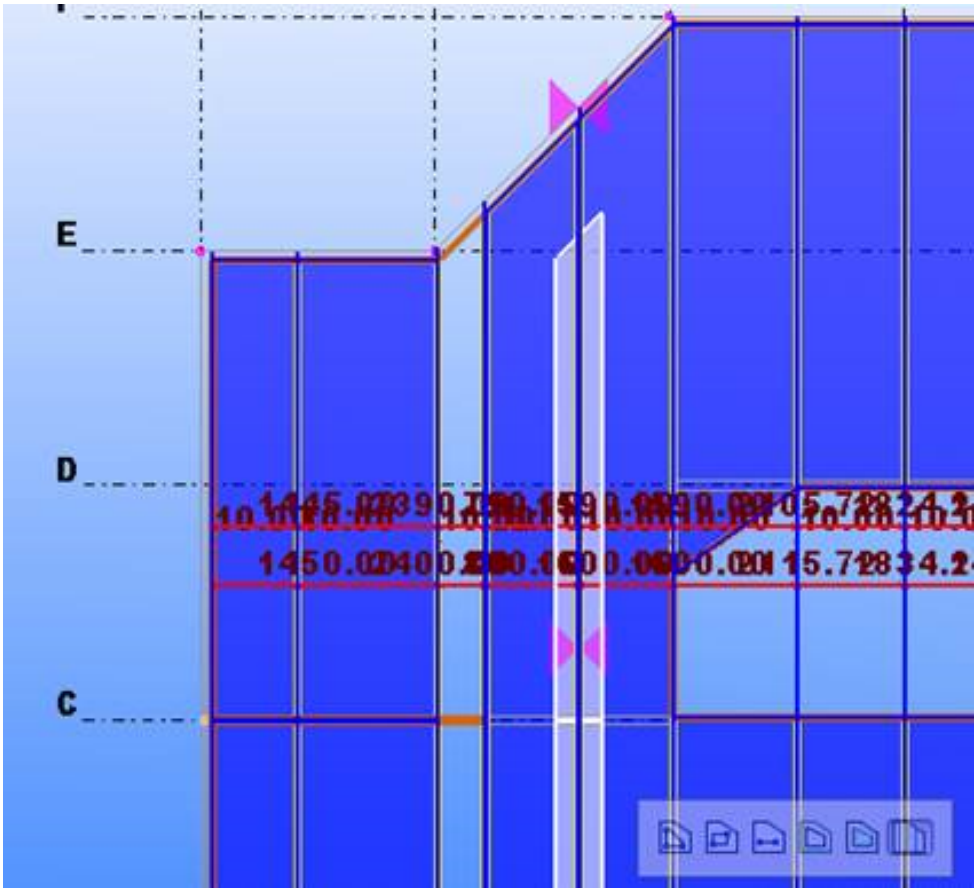
When the profile of the slab is fixed (not parametric), the slab is cut on either side when the width is smaller than the original width. The **Default** cut side is leading edge for the first slab and trailing edge for all others in relation to the layout direction.

You can modify the properties of multiple slabs at the same time by holding down the **Shift** key when selecting slabs.

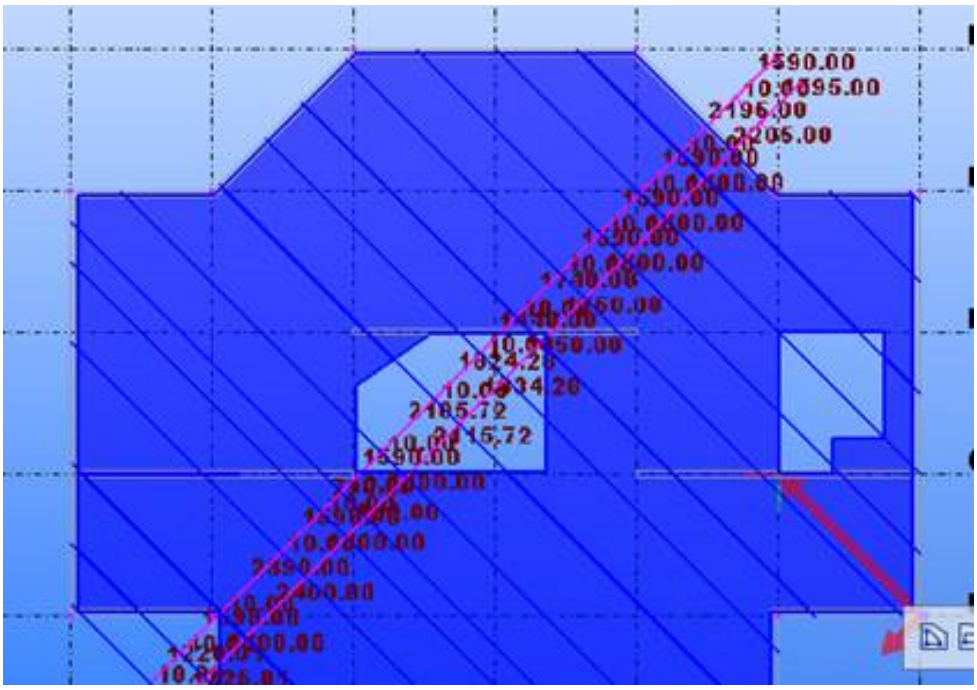
- To modify the gap width between the slabs, do one of the following:
  - Drag the end of the gap dimension.
  - Right-click a seam line handle, enter the width and click .

You can modify multiple gaps at same time by holding down the **Shift** key when selecting seam line handles.

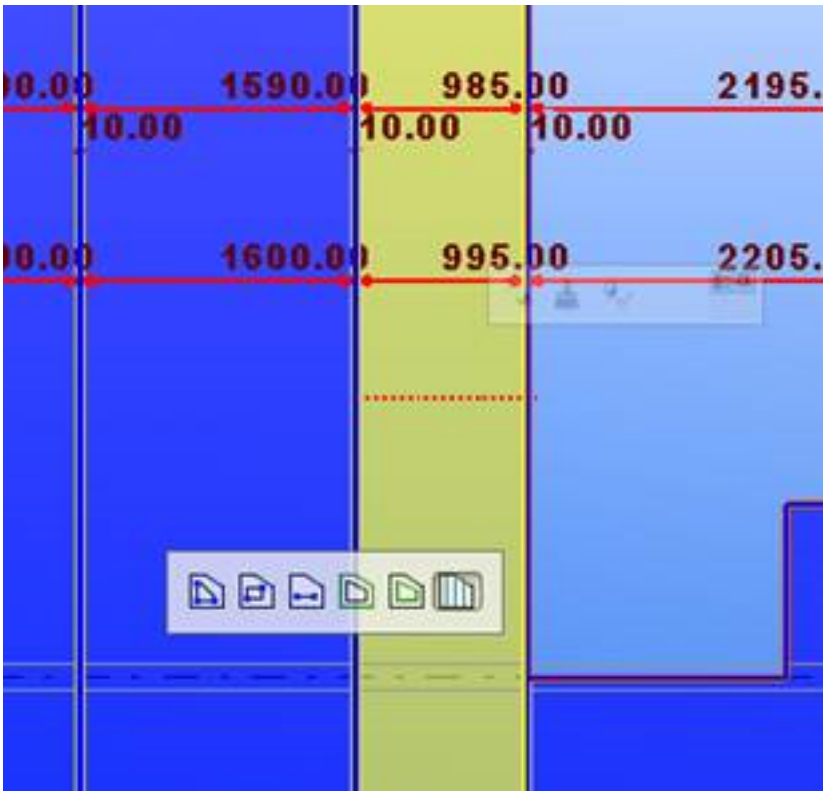
- To move a slab or change the order of slabs, drag the slab handles.



- To change the direction of slabs, drag the slab direction handle.

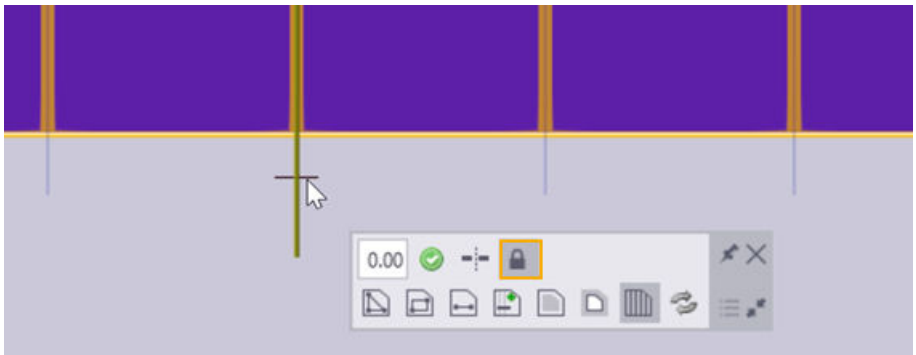


- To remove a slab from the layout, select the slab handles and press the **Delete** key.



Lock and unlock seams

You can lock and unlock seams on the contextual toolbar.



1. Select a seam.
2. Click to lock the seam.

   The button changes to when the seam is locked. Click the button again to unlock.

When a seam is locked and you are dragging the seams (with or without the **Alt**/**Ctrl** key pressed), only the seams in the range between the locked seams are changed. The first and last seams in the layout are always locked.

Locked seams keep their location, and new slabs are created or removed inside the locked range if you change the slab width, seam gap or seam center measures, or remove any of the slabs with narrowed width.

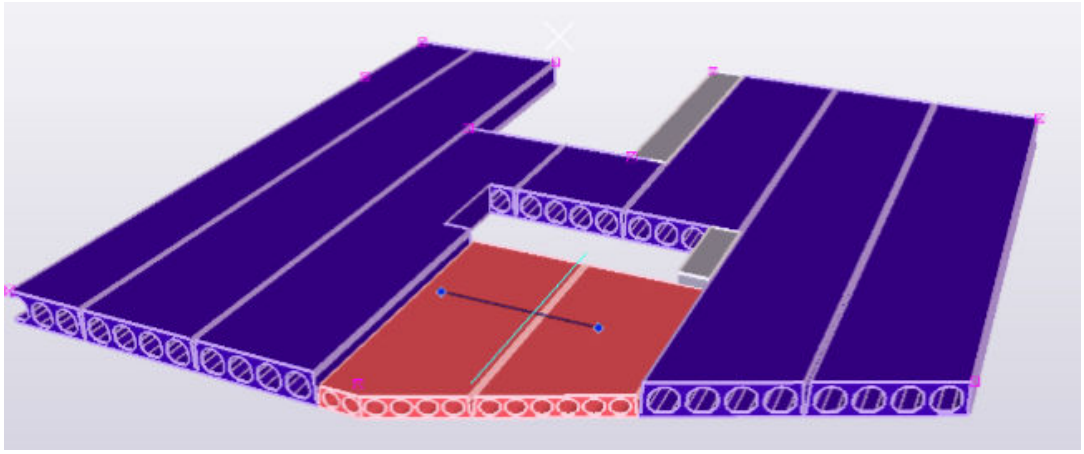
Add and modify property strips
You can use property strips to control the properties and detailing components of indivual slabs in the floor layout.

Property strips

You can define property strips that have certain properties and detailing components, and use the property strips according to your needs in the floor layout. For example, you can change the profile and class of the individual hollow core slabs. Note that property strips only affect the main layer parts.

Property strips are added as lines in the floor layout plane. Property strips affect the parts that they are touching. You can also set property strips so that they affect through the slab line.

The image below shows an example of a property strip. The two slabs in the floor layout have a different class and profile because of the property strip.



To create a property strip in the floor layout:

1. On the contextual toolbar of **Floor layout**, click to open a dialog box for defining the property strip.

2. On the **Properties** tab, define the individual slab properties.

Option	Description
Effect in layout	Select how the parts in the floor layout are affected: • **Hit slabs only**: The property strip only affects the parts it is touching. • **All slabs in line**: All slab parts in the line are affected if one slab line is split to two or more individual parts.
Layer thickness or profile	Change the thickness or profile of the layer.
Depth position	If the profile height is different, you can adjust the alignment according to: • **Bottom** face aligned with other slabs. • **Top** face aligned with other slabs.
Part name **Class** **Material** **Pour phase** **Part prefix** **Cast unit prefix** **Start no**	Define the name, class, material, part prefix and start number of the layer parts. Define the pour phase for CIP layers.

3. On the **Detailing** tab, select the detailing component that you want to use.

Option	Description
Component name	Select the detailing component from the list. The list of detailing components shows the components that you have added on the **Detailing** tab in the **Floor layout** dialog box. The content of the list may vary depending on your environment. You can also add new components to the list by selecting the **Add new**

Option	Description
	option. Note that you can only add detailing components that use one input part.
Component attr file name	Select the detailing component attributes that are used for the slab. Note that the content of the list varies depending on the component that you have selected.

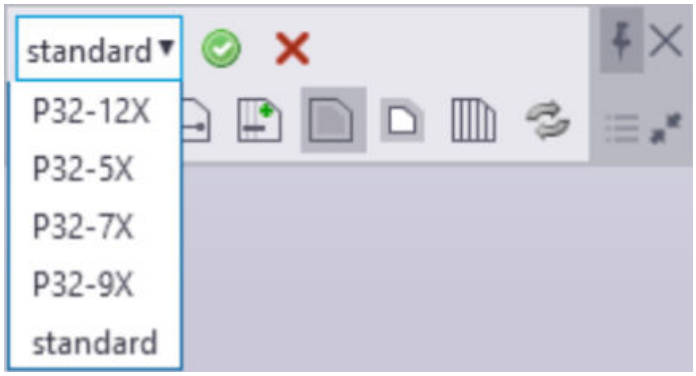
4. On the **User defined** tab, set the user-defined attribute values for the slabs.

   If you have defined the user-defined attributes in the part properties dialog box and want to use those values, leave the **User defined** tab empty.

5. Pick the start and end points for the property strip in the floor layout.

Modify the property strip

You can also use saved attribute settings from the contextual toolbar.



Use the and buttons on the toolbar as follows:

	Modify the detailing component and/or attribute file of the property strip. Select the property strip and select the component name and/or attribute file from the list. Click the button to modify the selected property strips. You can select multiple strips by holding down the **Shift** key.

	Delete a property strip. Select the strip that you want to delete. You can select multiple strips by holding down the **Shift** key.

You can also add new property strips by dragging an existing strip and holding down the **Ctrl** key.

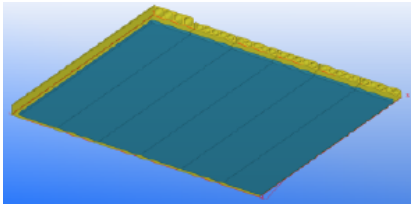
Floor Tool

Floor Tool creates a floor bay with optional insulation.

Objects created

- Hollow core slabs

Use for

Situation	Description
	Floor bay with insulation

Selection order

Make sure that you have defined the slab properties.

1. Pick the contour points of the floor bay.

   The floor bay is automatically created when you close the polygon.

NOTE The list of picked contour points on the **Bay contour** tab and the list of slabs on the **Slab list** tab are created when the slabs are created.

Slab properties tab

Use the **Slab properties** tab to control the profile, material and location of the hollow core slabs.

Slab attributes

Option	Description
Profile	Define the profile by selecting it from the profile catalog.
Name	Define a name for the slab.

Option	Description
Material	Define the material grade by selecting it from the material catalog.
Class	Define the part class number.
Prefix	Define the part prefix.
Start number	Define the start number for the cast unit.
Cut slab ends	Define whether the slab ends are cut or not. If you select **Yes**, define the minimum end angle for the slabs.
Rotation	Select an option to rotate the slab.
Vertical location	Define the offset location in the vertical direction.
Offset	Define the offset in the vertical direction.
Minimum end angle	Define the minimum end angle for the slabs. If the line to which the slab ends has a bigger angle than the defined minimum end angle, the part will be aligned to the line.

Additional attributes

Option	Description
Exposure Class	Define the exposure class for the hollow core slab. The exposure class will be saved as a user-defined attribute.
Relative strength	Define the relative strength of the hollow core slab. The relative strength will be saved as a user-defined attribute.
Fire rating	Define the fire rating for the hollow core slab. The fire rating will be saved as a user-defined attribute.
User-defined attributes	User-defined attributes are located in the `FloorTool.ini` file.

Slab direction

Option	Description
Slab direction	Define the direction of the slab. If you select **Angle**, you need to pick points in the model to define the new direction. The angle you have picked is shown in the **Angle**

Option	Description
	field. You can also define the angle directly in the field to rotate the slabs. Note that picking two points does not change the local +X direction.

Insulation tab
Use the **Insulation** tab to control the insulation properties of the hollow core slabs.

First define whether to create insulation or not. Insulation is created separately for each slab.

Insulation

Option	Description
Name	Define a name for the insulation.
Material	Define a material grade by selecting it from the material catalog.
Thickness	Define the thickness of the insulation.
Class	Define the part class number.
Offset at slab start/end	Define the offset between the slab start/end positions and the corresponding positions in the insulation parts.
Left/Right offset	Define the offset between the slab and the insulation for the rightmost and leftmost slab.

Bay contour tab
Use the **Bay contour** tab to control the coordinates of the picked contour points and the horizontal offset of the lines connecting the contour points. If the coordinates are not visible on the tab, click the **Get** button.

You need to create the hollow core slabs before using the **Bay contour** tab.

Bay contour properties

Option	Description
XY-constant Z	Define the Z coordinate for all contour points at the same specific height.
XYZ (sloping floor)	Define the Z constant for each contour point separately. The floor may be sloped.
Offset method	Define the offset method.
Default end offset	Define the default end offset.

Option	Description
Default side offset	Define the default side offset.

Bay contour table

The table lists the contour points showing their X, Y, and Z coordinates. You can modify the Z coordinate in the table only if you have selected **XYZ (sloping floor)**. You can modify the **Horizontal offset** on all rows of the table.

The table also shows the lines connecting the points. You can only modify the **Horizontal offset**. Entering a positive value enlarges the slab area and entering a negative value shrinks the slab area.

Slab list tab

Use the **Slab list** tab to list all the hollow core slabs created. If the slab list is not visible on the tab, click the **Get** button.

Slab properties

Option	Description
Slab number	Shows the order of the hollow core slabs starting from the first picked point.
Width	Shows the width of a narrowed slab.
Narrowed from	Define whether the slab is narrowed. The options are: • not narrowed (default) • **Right side** • **Left side**
Type	Define the slab type. • Use **Slab** for normal hollow core slabs. • Use **PIP** for a precast infill piece. • Use **GAP** for an empty space between the slabs. You do not need to define a profile when you select this option. • Use **CIP** to create a concrete part using the width defined in the **Width** column. The width range that you can use is defined in the `FloorTool.ini` file. The slabs listed in the `FloorTool.ini` file can only have a width that is inside the defined range. The slabs that are not listed can have any width. The value you give is rounded down to the nearest allowed value.

Option	Description
Profile	Define the profile by selecting it from the profile catalog.

You can **Add** and **Delete** slabs, and move them **Up** and **Down** in the list by using the buttons on the left. To restore the default values, click **Restore defaults**.

Concrete stairs

Tekla Structures includes the following tools you can use to create concrete stairs and stairwells:

- Concrete stairs (65) (page 3759)
- Stairwells and elevator shafts (90) (page 3774)
- Reinforced concrete stair (95) (page 3777)

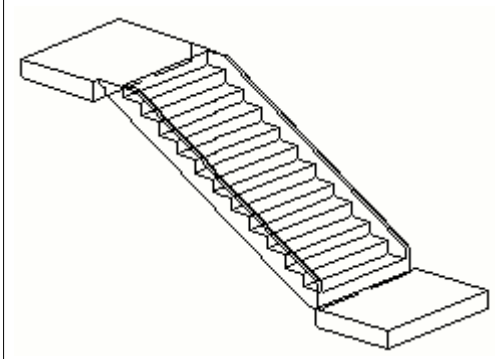
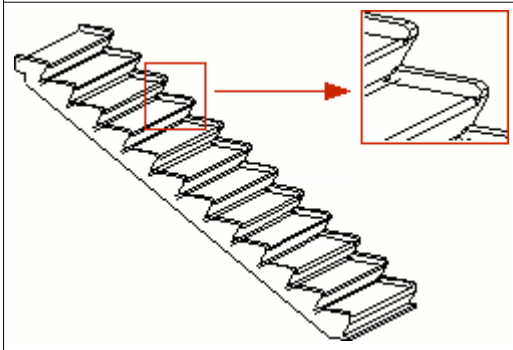
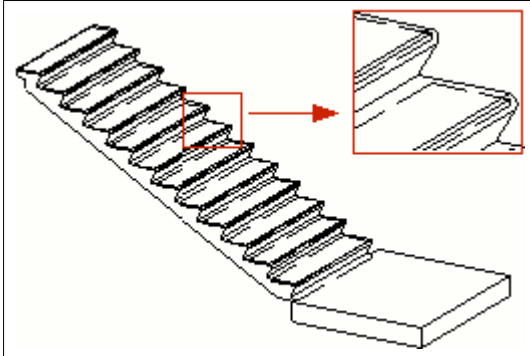
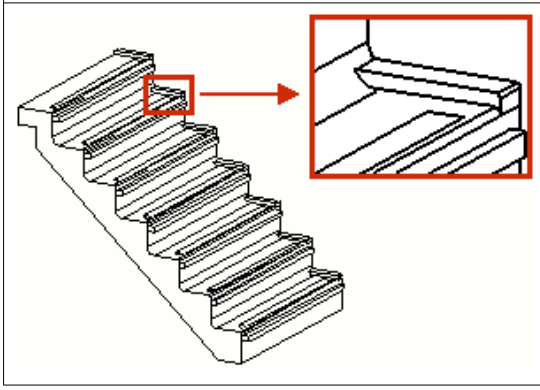
Concrete stairs (65)

Concrete stairs (65) creates precast stairs with optional landings, ridges, stringers, and anti-skid and kick plate profiles. It contains five different step shapes and options to create round or chamfered step edges.

Objects created

- Stairs
- Landings
- Ridges
- Stringers
- Anti-skid
- Kick plate

Use for

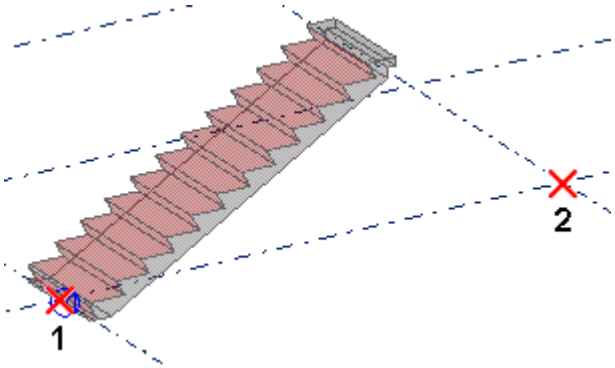
Situation	Description
	Staircase with • square step shape • stringers on both sides • top and bottom landings
	Staircase with • chamfered step corners • ridges on both sides
	Staircase with • rounded step corners • anti-skid profile • bottom landing
	Staircase with • square step shape • rounded step corners • ridges on both sides • anti-skid and kick plate profiles

Selection order

Selection order depends on the selected **Creation method** option on the **Stairs** tab.

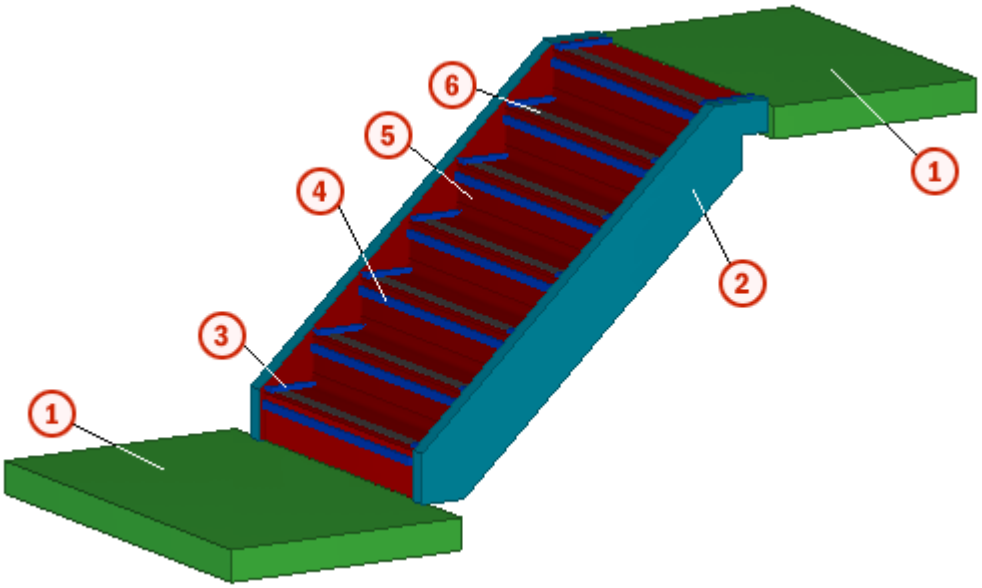
When **Creation method** is set to **Default**:

1. Pick the first point to indicate the start point of stairs.
2. Pick the second point to indicate the direction of stairs.



The stairs are created automatically.

Part identification key



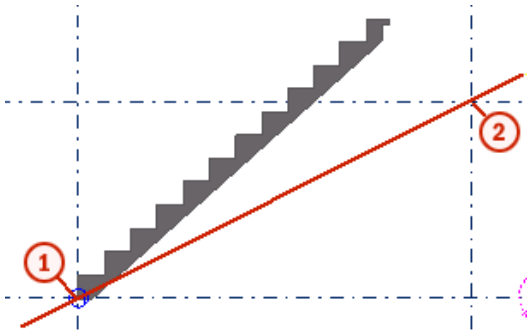
	Part
1	Landings
2	Stringers
3	Ridges
4	Kick plates
5	Steps

	Part
6	Anti-skids

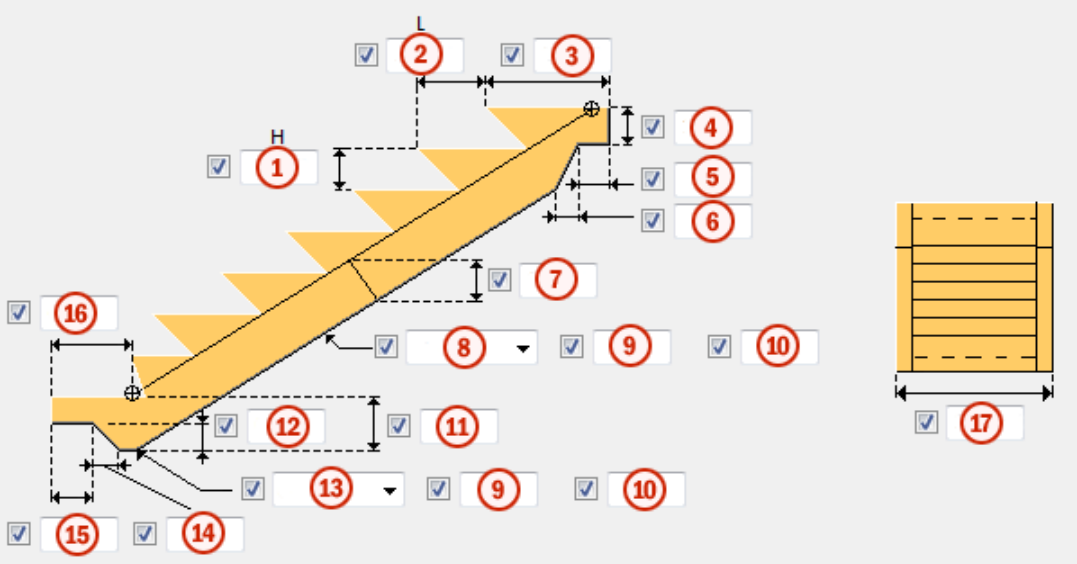
Stairs tab
Use the **Stairs** tab to define how the stairs are created and what are the dimensions of the stairs.

Creation method

Option	Description
Creation method	Select the creation method of the stairs. **Creation method** defines how many and which points you need to pick when you create the stairs. • **Default** Number of steps is calculated automatically using the vertical distance between the two input points. • **One point - N & L & H**: define N, L, and H (number, length, and height of the steps). • **Two points - N & L**: define N and L. Height of the steps is calculated automatically using the vertical distance between the two input points. • **Two points - N & H**: define N and H. Length of the steps is calculated automatically using the horizontal distance between the two input points. • **Two points - L & H**: define L and H. Number of steps is calculated automatically using the vertical distance between the two input points. • **Two points - N**: define N. Height and length of steps is calculated automatically using the

Option	Description
	horizontal and vertical distance between the two input points. You can define the length of the top step.
No of steps, N	Define the number of steps, if the **Two points - L & H** option is not selected.
Position	Define the horizontal position of the staircase, relative to the line joining the picked points.
Distance	Enter a distance to move the stairs in the horizontal direction from the line joining the picked points. The selected **Position** option has an effect on this dimension.
Rotation	Define the angle between the line joining the steps and the line joining the selected points. 
Cast unit type	Select the cast unit type: • **Precast** • **Cast in place**

Stair dimensions



	Description
1	Height of the step.
2	Length of the step.
3	Length of the top step.
4	Height of the top step.
5	Width of the top corbel.
6	Slope from the corbel to the underside of the stairs.
7	Thickness of the slab.
8	Chamfering for the underside of stairs.
9 **10**	Chamfer or radius values.
11	Height of the bottom step.
12	Height of the stair foot.
13	Chamfering for the bottom of stairs.
14	Slope from the foot nose to the foot base.
15	Length of the bottom cut.
16	Length of the stair foot.
17	Width of the staircase, including stringers.

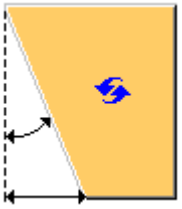
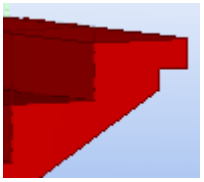
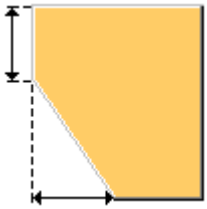
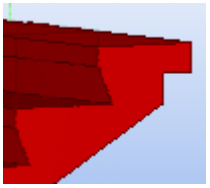
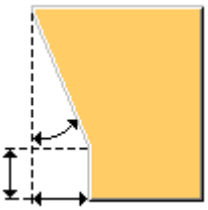
Create foot

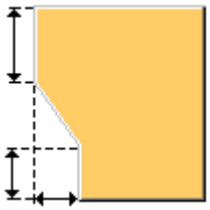
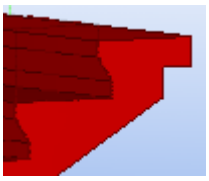
Define whether a foot is created at the bottom of the stairs.

Step shapes tab
Use the **Step shapes** tab to define the shape of steps.

All other steps/Last step at foot

Select the step shape. You can define the shape of the last step at the bottom if you set **Last step same as other step** to **No**.

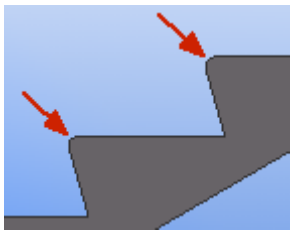
Option	Description
	Default Beveled From the list below the step shape options, select whether you want to enter a dimension or an angle. AutoDefaults can change this option.
	Straight step 
	Top straight, bottom beveled Enter two dimensions. 
	Top beveled, bottom straight Enter the vertical dimension and then select whether you want to enter an angle or a horizontal dimension.

Option	Description
	Top and bottom staight Enter vertical dimensions and then select whether you want to enter an angle or a horizontal dimension. 

out

Select the shape of the step front edge and enter the required dimensions.

- **Default**
- **Radius**: Creates rounded edge.

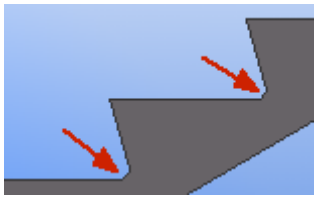


- **Chamfer**: Creates chamfered edge.
- **No**: Creates square edge.

in

Select the shape of the step inner corner and enter the required dimensions.

- **Default**
- **Radius**: Creates rounded inner corner.



- **Chamfer**: Creates chamfered inner corner.
- **No**: Creates square inner corner.

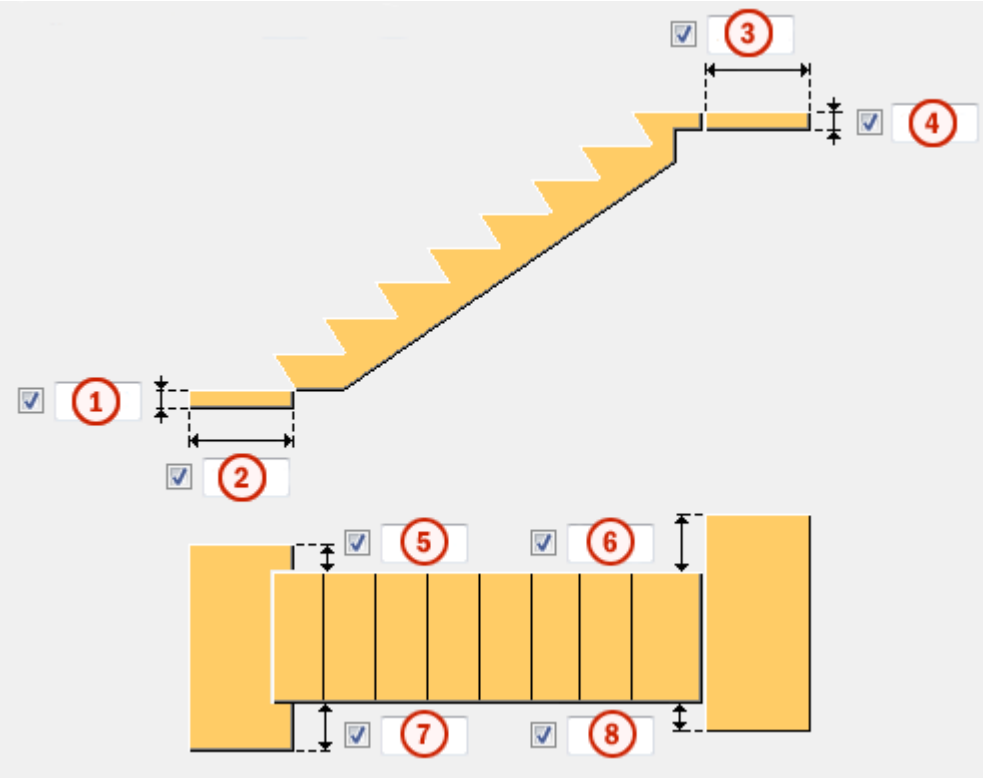
Landings tab

Use the **Landings** tab to create top and/or bottom landings.

Create landings

Define whether one or two landings are created, or whether there are no landings.

Landing dimensions and extensions

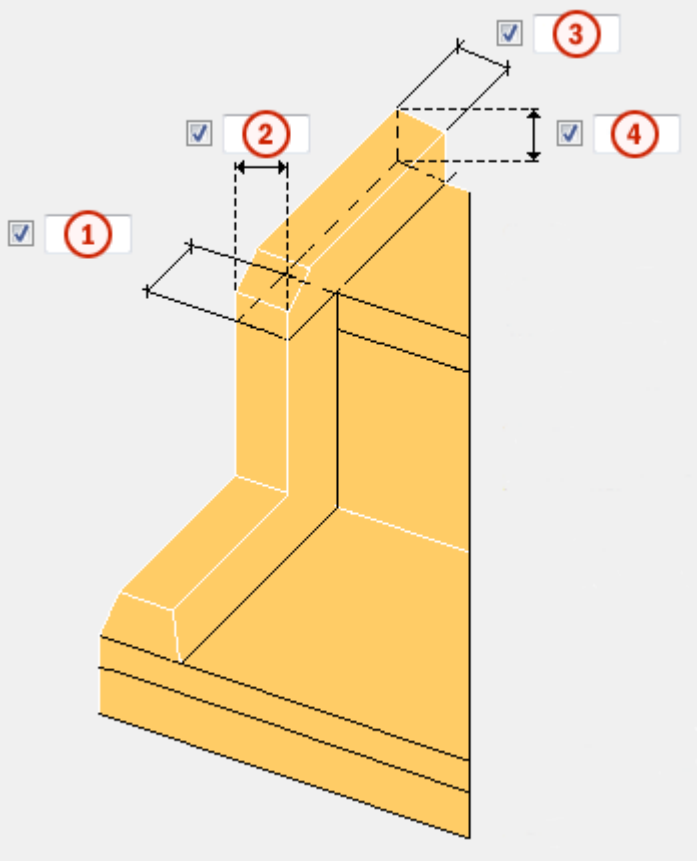


	Description
1	Thickness of the bottom landing.
2	Length of the bottom landing.
3	Length of the top landing.
4	Thickness of the top landing.
5 **7**	Left and right extensions of the bottom landing.
6 **8**	Left and right extensions of the top landing.

Ridges tab

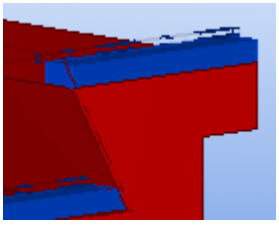
Use the **Ridges** tab to create horizontal and/or vertical ridges on both sides or only on the other side of the stairs.

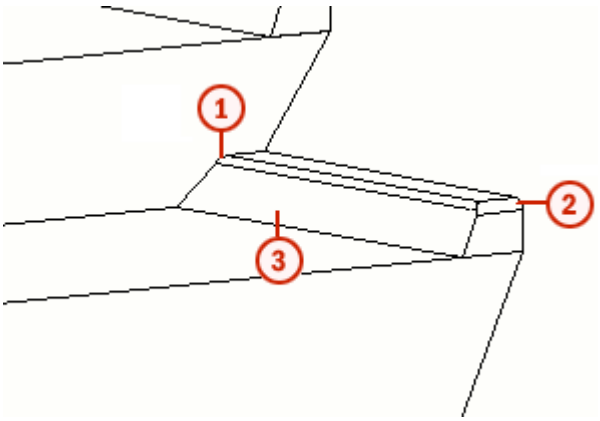
Dimensions



	Description
1	Thickness of the vertical ridge.
2	Width of the vertical ridge.
3	Width of the horizontal ridge.
4	Thickness/height of the horizontal ridge.

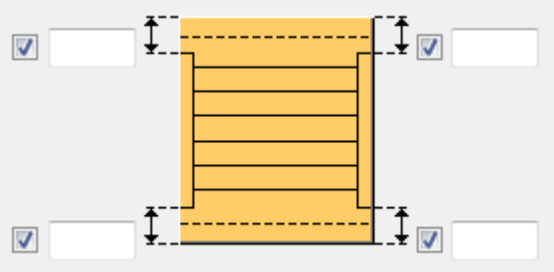
Horizontal ridges

Part	Description
Create	Define whether horizontal ridges are created. 

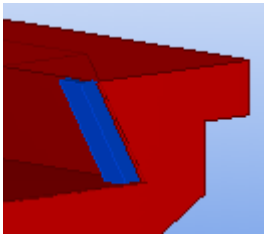
Part	Description
Create at foot	Define whether horizontal ridges are created at the foot of the stairs. This option works in the same way as the **Create** option.
	
Inside chamfer **1**	Select the type of the inside chamfer and enter the required dimensions.
Corner chamfer **2**	Select the type of the corner chamfer and enter the required dimensions.
Slope **3**	Set the slope as an angle or a dimension. The slope makes the ridge inclined.
Foot corner chamfer	Define whether corner chamfers are created at the stair foot.

Top and foot ridges

Define the horizontal ridge length at the stair top step and at the stair foot. When no foot is created, vertical ridges continue to the bottom level of the stairs.



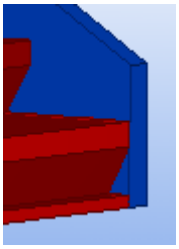
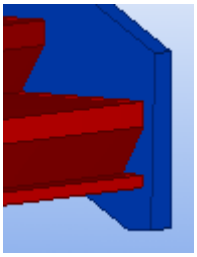
Vertical ridges

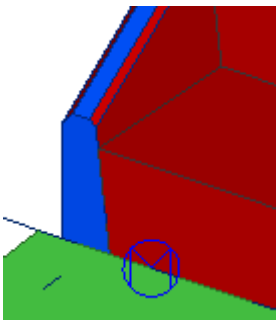
Part	Description
Create	Define whether vertical ridges are created. 
Slope	Set the slope as an angle or a dimension. The slope makes the ridge inclined.

Stringers tab

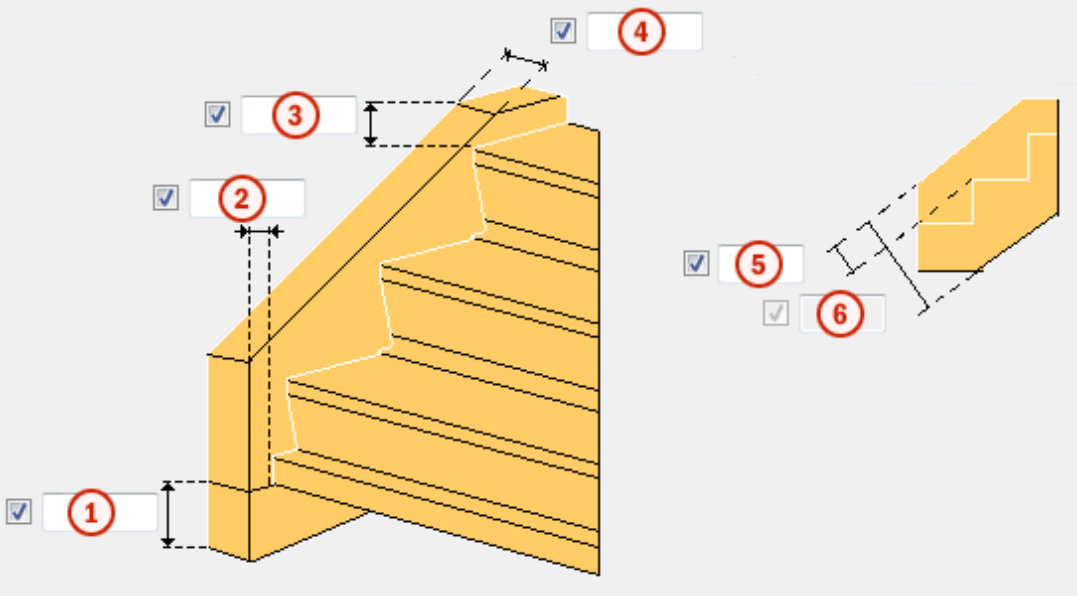
Use the **Stringers** tab to create stringers and ridges on the left, right, or both sides of the stairs.

Parts

Part	Description
Create stringers	Define whether stringers are created.
Create top ridge	Select an option to create top ridges. 
Create bottom ridge	Select an option to create bottom ridges. 

Part	Description
Slope	Use to incline the inner surface of the stringers.  You can define the slope using an angle or a dimension.
Inside chamfer	Define whether the inside edge of stringer is chamfered or not.
Outside chamfer	Define whether the outside edge of stringer is chamfered or not.
Stringer height	Define how the stringer height is determined.

Dimensions



	Description
1	Height of the stringer bottom ridge.
2	Horizontal offset of the bottom ridge from the bottom step.
3	Height of the stringer top ridge.

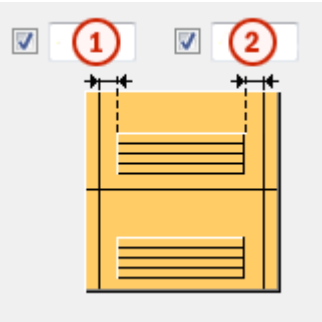
	Description
4	Width of the stringer.
5	Difference in height that is the perpendicular distance between the step edge and stringer edge. You can enter this dimension if **Stringer height** is **Diff in height**.
6	Total height of the stringer. You can enter this dimension if **Stringer height** is **Default** or **Total height**.

Anti-skid/Kick plate tab
Use the **Anti-skid** and **Kick plate** tabs to create slip resistant surfaces.

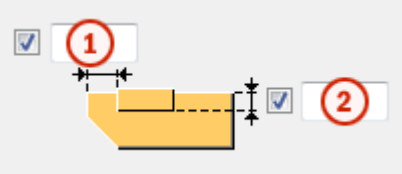
Parts

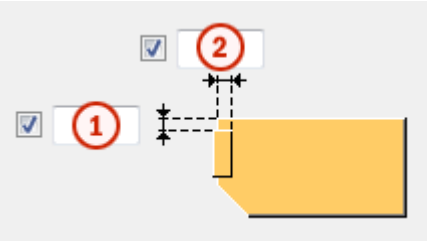
Part	Description
Creation anti-skid **Creation kick plate**	Define whether anti-skids or kick plates are created.
Create anti-skid on foot	Define whether anti-skids are created at the stair foot.
Create cutout	Define whether you want to create cutouts with the anti-skid profile. By default, the cutouts are not created.
Include in cast unit	Define whether anti-skids or kick plates are included in cast unit.
Profile	Define the anti-skid or kick plate profile by selecting it from the profile catalog.
Rotation	Select an option to rotate the anti-skid or kick plate profile.

Anti-skid/kick plate position



	Description
1	Anti-skid or kick plate distance from the left edge of the step.
2	Anti-skid or kick plate distance from the right edge of the step.





	Description
1	Distance from the step front edge.
2	Depth for the anti-skid or kick plate.

Attributes tab
Use the **Attributes** tab to control the properties of different stair parts.

Part attributes

Option	Description	Default
Pos_No	**Prefix** and **Start number** for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options** .
Material	Material grade.	The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options** .
Name	Name that is shown in drawings and reports.	
Class	Part class number.	

UDA tab
Use the **UDA** tab to provide additional information about the parts created.

TIP The user-defined attributes are case sensitive. Check the correct spelling of an attribute from the `objects.inp` file.

Stairwells and elevator shafts (90)

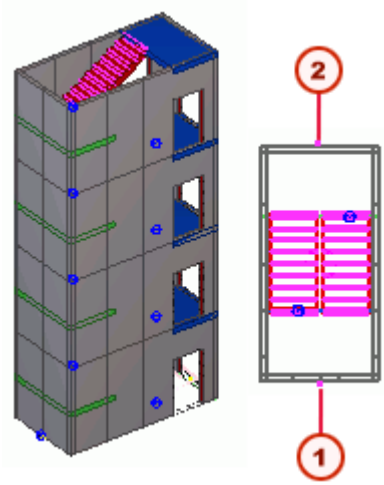
Stairwells and elevator shafts (90) creates a rectangular stairwell or elevator shaft to represent the structure in the design stage.

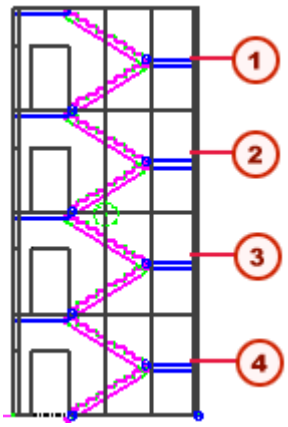
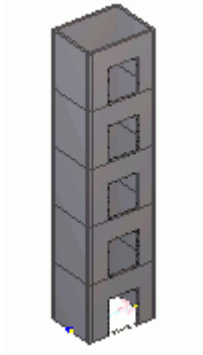
When you detail the structure, you can apply seams, connections, and reinforcement to complete the details of a stairwell or elevator shaft. This component uses the Concrete stairs (65) (page 3759) component for creating stairs.

Parts created

- Wall panels
- Landings (optional)
- Flights of stairs (optional)
- Roof slab (optional)
- Openings (optional)

Where to use

Situation	More information
	A stairwell over four floors created using the default options. As shown, the points picked and the dimensions on the **Stairs and landings** tab determine the length of the stairwell. Walls consist of one or more precast concrete panels. **1** First point picked **2** Second point picked

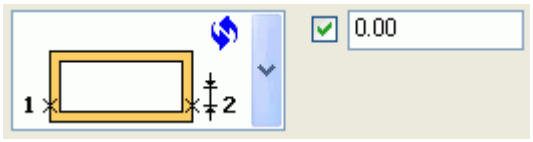
Situation	More information
	Stairwells always contain a first floor and a top floor. Indicate the number of additional floors on the **Plan view** tab, **Levels** field. This stairwell has 2 additional floors, for a total of 4 floors. **1** Top floor **2** Additional floor **3** Additional floor **4** 1st or ground floor
	Elevator shaft created by selecting the **Elevator shaft** option on the **Stairs and landings** tab, **Type** field, for all floors.

Limitations

The minimum floor height between floors is 200 mm.

Picking order

1. On the **Plan view** tab, check the **Position** option. The default is in the middle of walls 3 and 4.

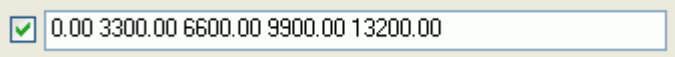


2. Pick a point to indicate the position of wall 3.
3. Pick a point to indicate the position of wall 4.

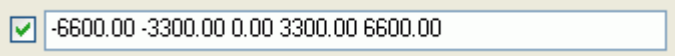
Plan view tab

Use the **Plan view** tab to define the properties of the stairwell or elevator shaft.

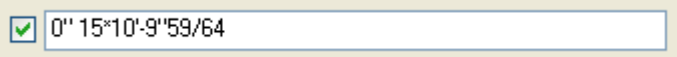
To define the number of levels and floor heights, enter a height for each floor level in the **Levels** field, for example:



You can also define negative floor levels, for example:



You can also use multiplier to define many floors, as in the following US imperial example:

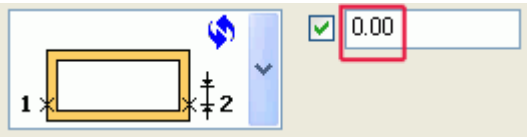


Tekla Structures also uses these values to calculate the total height of the stairwell.

To create the stairwell or elevator shaft, you pick two points in the model. Use the **Position** field to define the location of the stairwell or elevator shaft relative to these points. The options are:

- Middle (default)
- Left
- Right

Enter the offset distance from the selected position. Here, the offset is from the middle of wall.

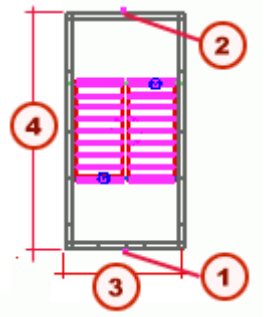


To manually define the total length and total width of the stairwell or elevator shaft, enter the following values:

L - total length

W - total width

Leave both fields blank to have Tekla Structures automatically calculate these dimensions from the the points picked:



	Description
1	First point picked.
2	Second point picked.
3	Total width, W
4	Total length, L

NOTE The dimensions defined on the **Stairs and landings** tab may override the total length or width that is automatically calculated or that you manually enter.

Openings tab

Use the **Openings** tab to define opening dimensions for each wall panel and for the first, additional, and top floors.

Each wall may have a single opening, or no opening.

Stairs and landings tab

Use the **Stairs and landings** tab to define the stair and landing properties and dimensions.

Select to create an elevator shaft from the **Type** list.

Attributes tab

Use the **Attributes** tab to define basic part properties for walls, stairs, and landings.

You can also use the save properties from the Concrete stairs (65) (page 3759) component for the first, additional, and top stairs.

Option	Description
Pos_No	To ensure correct numbering, define a **Prefix** and a **Start number** for the part position number of stair parts. Enter the assembly position number of the stair parts on the second row.
Material	Define the material grade.
Name	Define a name that is shown in drawings and reports.
Class	Define the part class number.

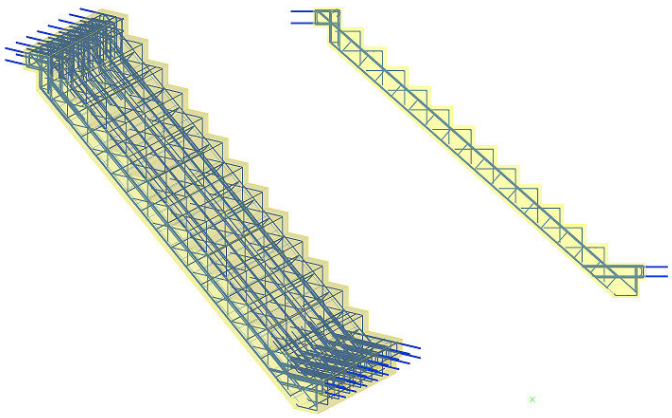
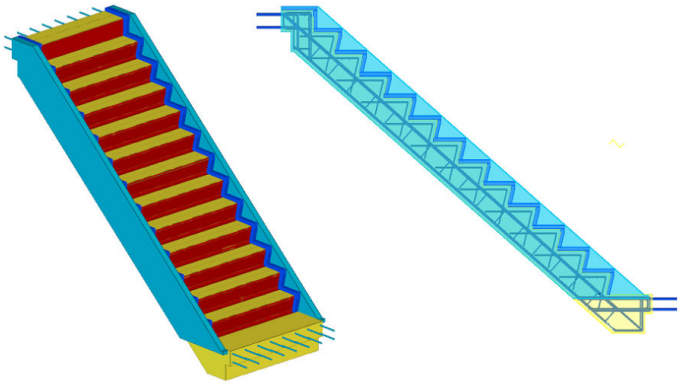
Reinforced concrete stair (95)

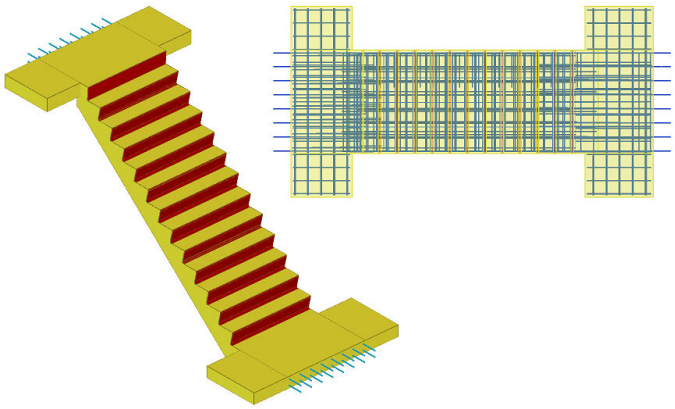
Reinforced Concrete Stair (95) creates reinforced concrete stairs.

Objects created

- Stairs
- Landings (optional)
- Ridges (optional)
- Stringers (optional)
- Anti-skid (optional)
- Main bars and stirrups of stair reinforcement (**Bar A** - **Bar L** tabs) (optional)
- Meshes in stairs and landings (optional)
- Mesh bars (optional)
- Anchor bars (optional)
- Landing end bars (optional)

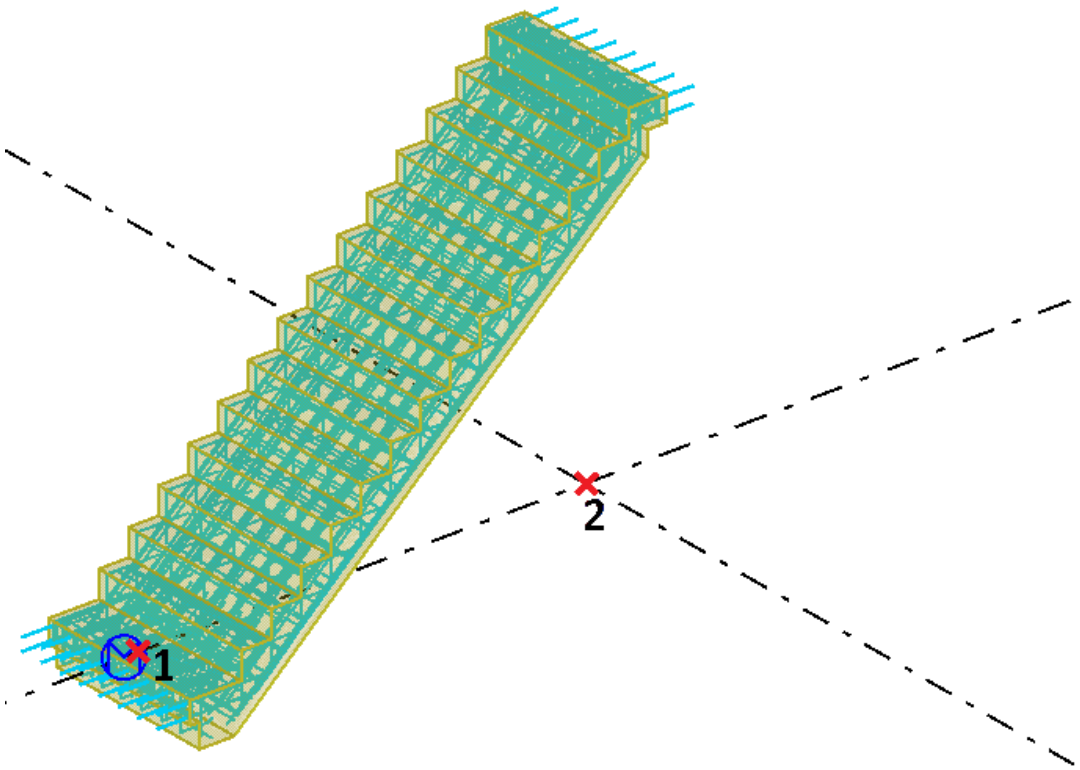
Use for

Situation	Description
	Reinforced concrete stairs.
	Reinforced concrete stairs with chamfered steps, ridges and stringers.

Situation	Description
	Reinforced concrete stairs with reinforced landings.

Selection order

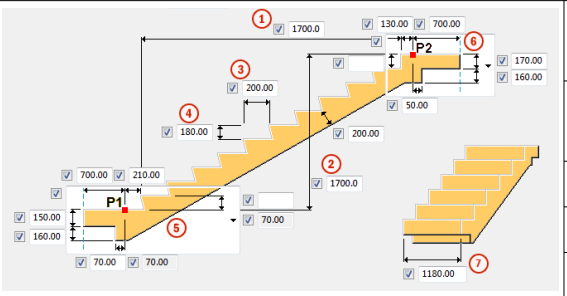
1. Pick the first point to indicate the start point of the stairs.
2. Pick the second point to indicate the direction of the stairs.
3. Select any number of parts to be cut by the stairs (optional).
4. Click the middle mouse button to create the stairs.



Parameters tab

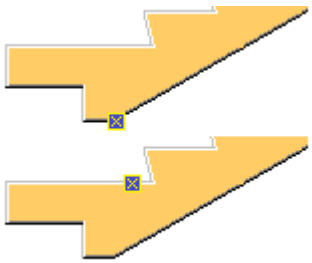
Use the **Parameters** tab to control the shape of the stairs, number of steps, stair creation method and the stairs width.

Stair dimensions

	Description
	
1	Horizontal dimension of the steps area
2	Vertical dimension of the steps area
3	Step length
4	Step height
5	Shape options for stair bottom
6	Shape options for stair top
7	Stair width

Start point position

Select the start point position of the polygon shape.



NOTE Start point position affects the bounding box of the cast-unit. Therefore it will affect drawing view orientation and numbering.

Stair creation method

Option	Description
P1 P2 step length	Create stairs between points P1 and P2. Stair dimension are defined by the distance between P1 and P2 and step length.
P1 P2 step height	Create stairs between points P1 and P2. Stair dimension are defined by the distance between P1 and P2 and step height.
P1 step length step height N steps	Create stairs from point P1 to point P2. Stair dimensions are defined by P1 and step height, length and number of steps.

Option	Description
P2 step length step height N steps	Create stairs from point P1 to point P2. Stair dimensions are defined by P2 and step height, length and number of steps.
P1 horizontal and vertical step distance	Create stairs from point P1 to point P2. Stair dimensions are defined by P1 and horizontal and vertical step distance.
P2 horizontal and vertical step distance	Create stairs from point P1 to point P2. Stair dimensions are defined by P2 and horizontal and vertical step distance.

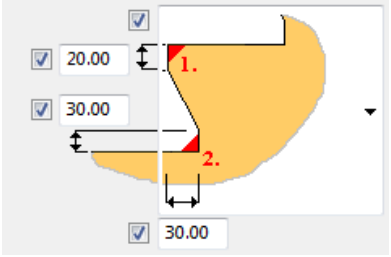
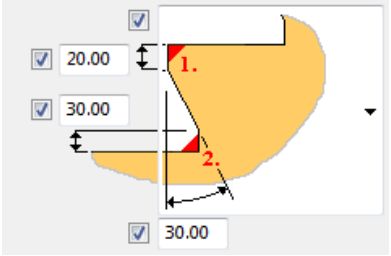
Parts tab

Use the **Parts** tab to control the material, name, class, positioning, cast unit type and step chamfering.

Step chamfer settings

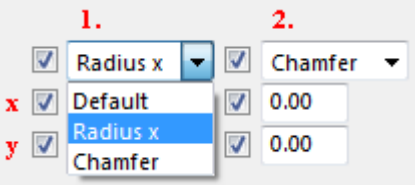
Option	Description
Middle for all	All steps are chamfered according to the **Middle steps** settings.
Bottom different	Bottom step is chamfered according to the **1st step at bottom** settings. All other steps are chamfered according to the **Middle steps** settings.
Top different	Top step is chamfered according to the **Last top step** settings. All other steps are chamfered according to the **Middle steps** settings.
Bottom and top different	Bottom step is chamfered according to the **1st step at bottom** settings. Top step chamfered are according to the **Last top step** settings. All other steps are chamfered according to the **Middle steps** settings.

Step chamfer settings are set in the same way for **1st step at bottom**, **Middle steps** and **Last top step**.

Option	Description
	Size of step cut is defined by the distance.
	Size of step cut is defined by the angle.

Corner chamfer types

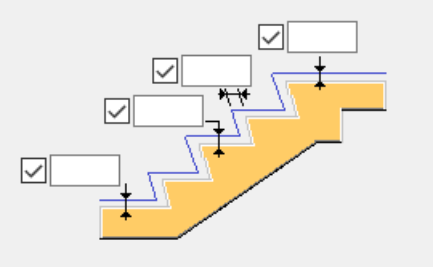
Corner chamfers **1.** and **2.** can be defined by the **Radius x** or by the sides of the **Chamfer x, y**.



Create finished floor level

Select whether to create finished floor level. The finished floor level is defined by a polygon created from an offset stairs polygon and it is used for drawings.

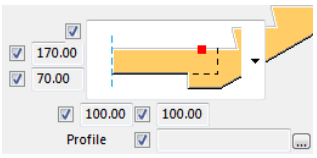
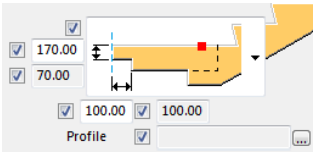
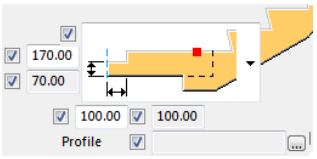
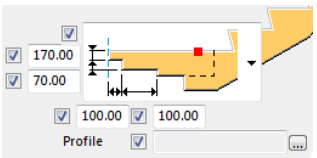
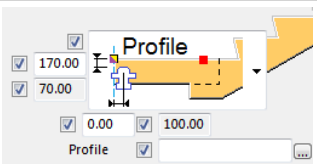
Define the profile, properties and offsets of the dummy part that is created at each polygon vertex, and the position in depth. You can define different offsets for floor level profiles for the steps, and the top and bottom landing, and the vertical cover thickness.



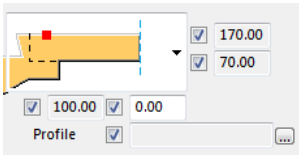
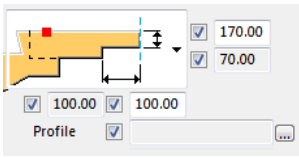
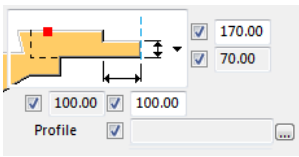
Stairs and landings tab

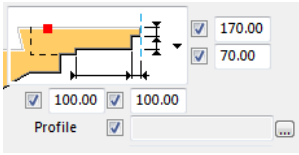
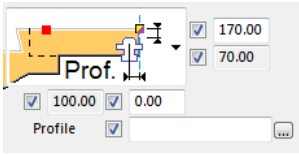
Use the **Stairs and landings** tab to control the size and type of the bottom landing and the top landing.

Bottom landing cut option

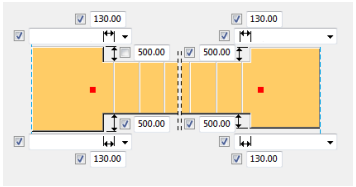
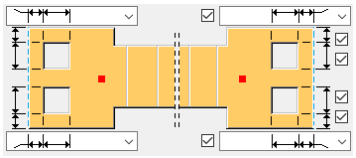
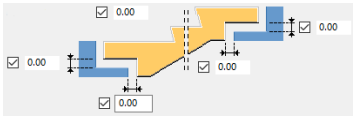
Option	Description
	Bottom landing with no cut.
	Bottom landing with cut defined by its length and distance from the top side of landing.
	Bottom landing with cut defined by its length and distance from the bottom side of landing.
	Bottom landing with L shape cut defined by its three dimensions and distance from the top side of landing.
	Bottom landing cut defined by a profile and its position in landing.

Top landing cut option

Option	Description
	Top landing with no cut.
	Top landing with cut defined by its length and distance from the top side of landing.
	Top landing with cut defined by its length and distance from the bottom side of landing.

Option	Description
	Top landing with L shape cut defined by its three dimensions and distance from the top side of landing.
	Top landing cut defined by a profile and its position in landing.

Landing dimensioning and openings

	Description
	Use the top view setting to define the dimensions of the bottom landing and the top landing.
	Use the top view setting to define two openings for the bottom and the top landing. The following rebars avoid the created openings: anchor and Z anchor bars, mesh bars, landing end bars, and A, B, C, E, G, K bars.
Extra landings	Define an individual name and class for the extra landing parts.
	Define the gaps on the sides of the landings.

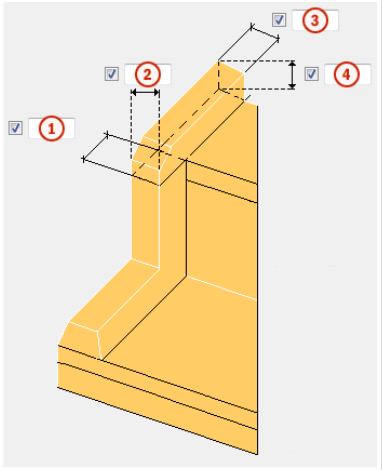
Recesses and holes

Option	Description
	No hole or recess
	Hole
	Recess at the top face
	Recess at the bottom face

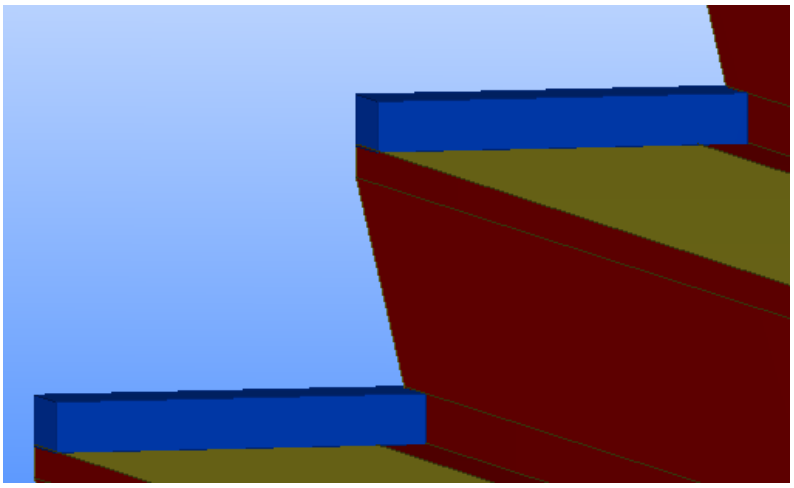
Ridges tab

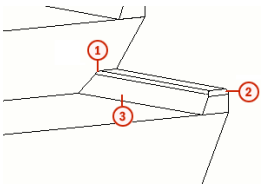
Use the **Ridges** tab to create horizontal and/or vertical ridges on both sides or only on one side of the stairs.

Dimensions

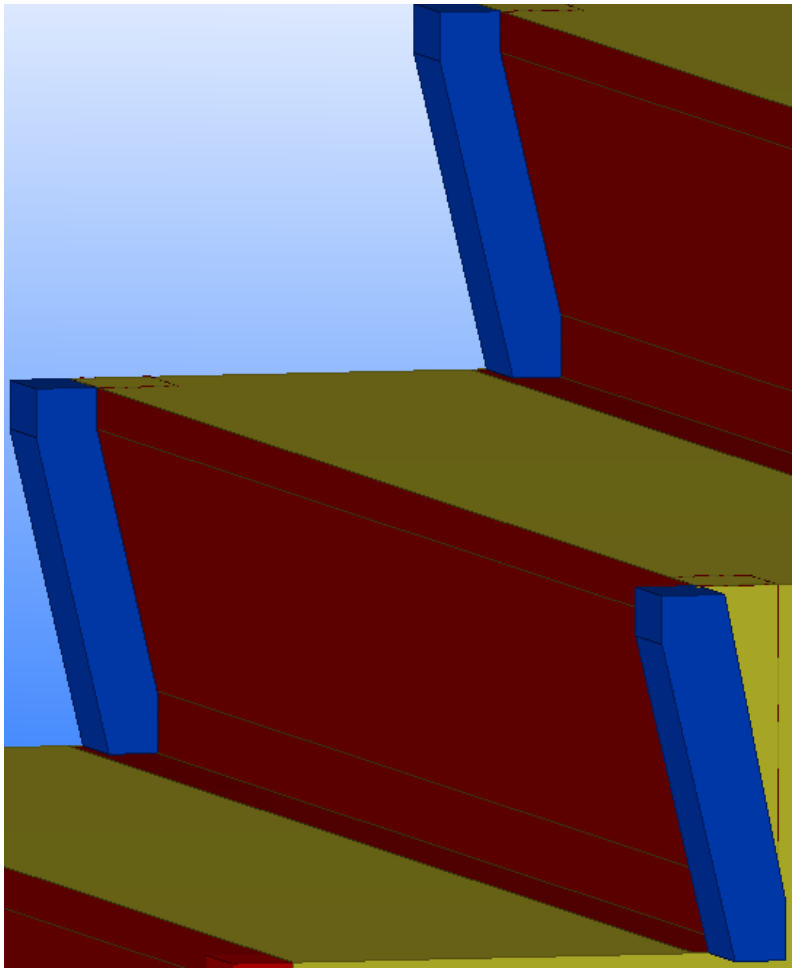
		Description
	**1**	Thickness of the vertical ridge.
	2	Width of the vertical ridge.
	3	Width of the horizontal ridge.
	4	Thickness/height of the horizontal ridge.

Horizontal ridges

Option	Description
Create	Define whether horizontal ridges are created. 
Create at foot	Define whether horizontal ridges are created at the foot of the stairs. This option works in the same way as the **Create** option.

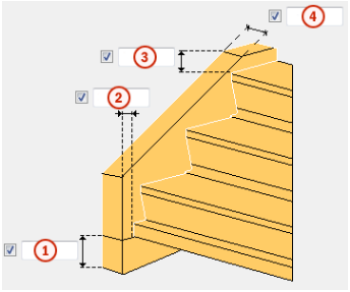
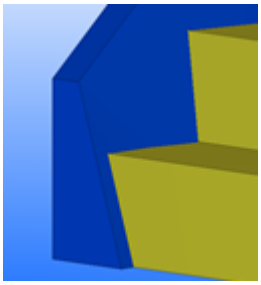
Option	Description
Chamfering  	**1 Inside chamfer**: Select the type of the inside chamfer and enter the required dimensions. **2 Corner chamfer**: Select the type of the corner chamfer and enter the required dimensions. **3 Slope**: Set the slope as an angle or a dimension. The slope makes the ridge inclined.
Foot corner chamfer	Define whether corner chamfers are created at the stair foot.

Vertical ridges

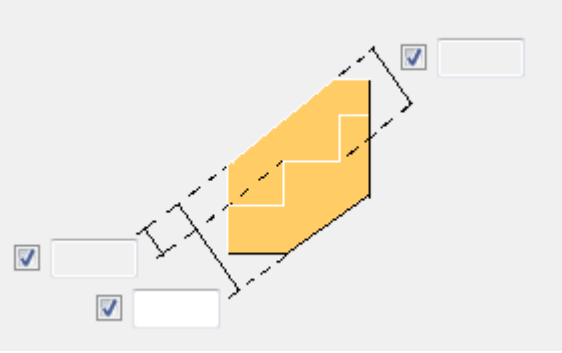
Option	Description
Create	Define whether vertical ridges are created. 
Slope	Set the slope as an angle or a dimension. The slope makes the ridge inclined.

Stringers tab

Use the **Stringers** tab to create the stringers and ridges on the left, right, or both sides of the stairs.

		Description
	**1**	Height of the stringer bottom ridge.
	2	Horizontal offset of the bottom ridge from the bottom step.
	3	Height of the stringer top ridge.
	4	Width of the stringer.
Slope		Use to incline the inner surface of the stringers.  You can define the slope using an angle or a dimension.
Inside chamfer		Define whether the inside edge of stringer is chamfered or not.
Outside chamfer		Define whether the outside edge of stringer is chamfered or not.

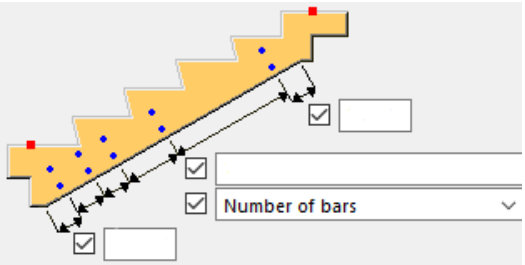
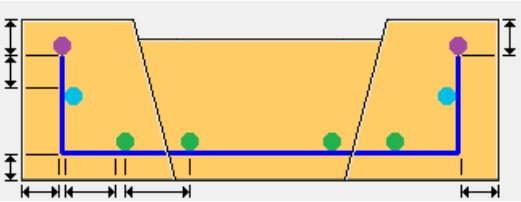
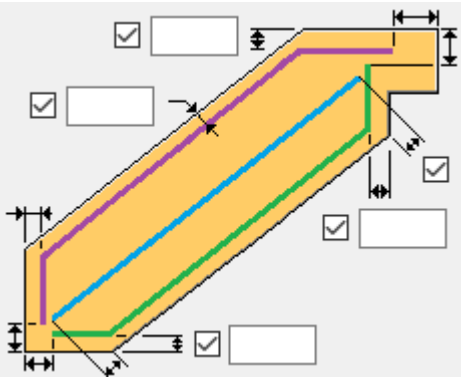
Stringer height



Stringer height based on the total height or distance from inner/outer step corner.

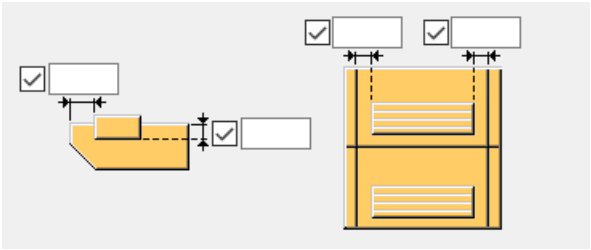
Stringer bars tab

Use the **Stringer bars** tab to create reinforcements in the stringers.

Option	Description
  	Define the spacing of bars and the concrete cover thicknesses.
Create	Select to create the group of bars.
Grade	Strength of the steel used in the reinforcing bars.
Size	Diameter of the reinforcing bar.
Radius	Internal radius of the bends in the bar.
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Class	Use **Class** to group reinforcement.
Name	Tekla Structures uses the name in drawings and reports.

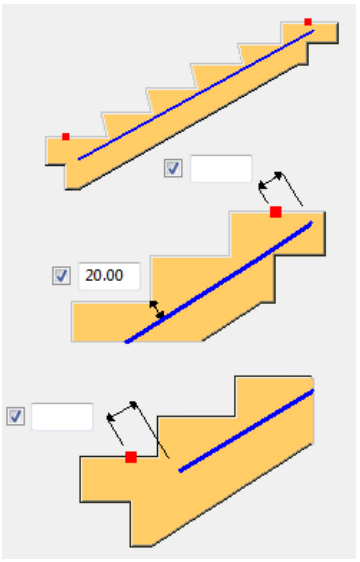
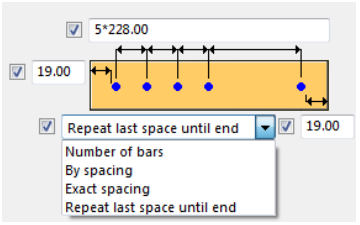
Anti-skid tab

Use the **Anti-skid** tab to create slip resistant surfaces.

Option	Description
Creation anti-skid	Define whether anti-skids are created.
First bottom anti-skid	Define the material, name and class of the anti-skid profile at the bottom.
Last top anti-skid	Define the material, name and class of the anti-skid profile at the top.
Create anti-skid on foot/top	Define whether anti-skids are created at the stair foot/top.
Create cutout	Define whether you want to create cutouts with the anti-skid profile. By default, the cutouts are not created.
Include in cast unit	Define whether anti-skids are included in the cast unit.
Profile	Define the anti-skid profile by selecting it from the profile catalog.
Rotation	Select an option to rotate the anti-skid profile.
Offsets	Define the anti-skid profile offsets on the steps. 

Bar A tab

Use the **Bar A** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group.

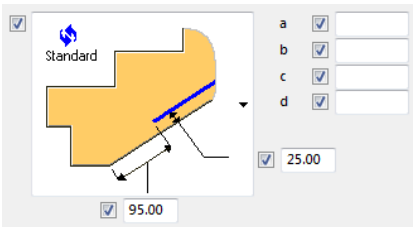
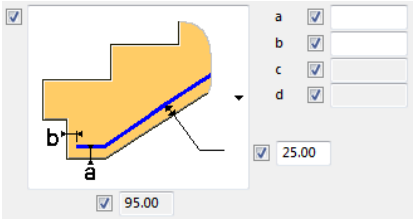
Option	Description
	Geometry and concrete cover thickness.
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

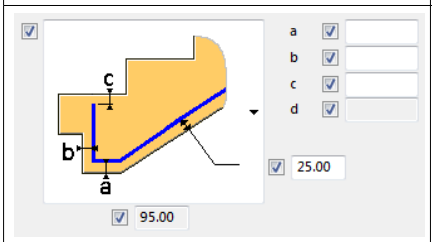
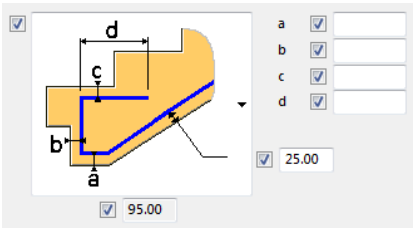
Bar B tab

Use the Bar B tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group.

Bar B bottom ending options

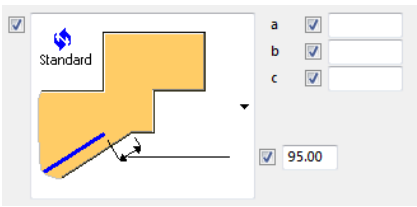
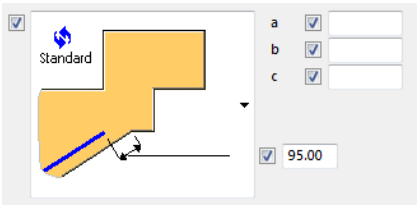
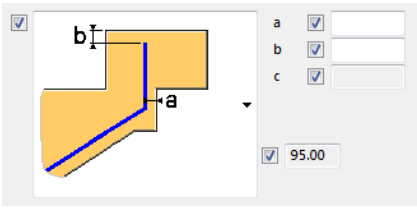
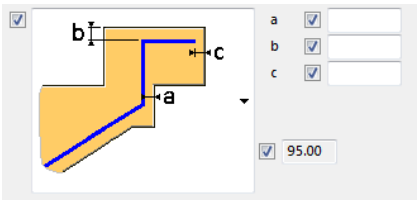
Define geometry and concrete cover thickness of the bar B bottom ending.

Option	Description
	Bar B simple bottom ending.
	Bar B bottom ending bent once to fit the bottom footing of the stairs.

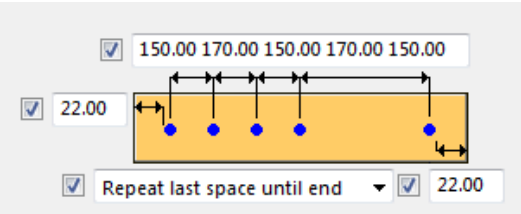
Option	Description
	Bar B bottom ending bent twice to fit the bottom footing of the stairs.
	Bar B bottom ending bent up to three times to fit the bottom footing of the stairs.

Bar B top ending options

Define geometry and concrete cover thickness of the bar B top ending.

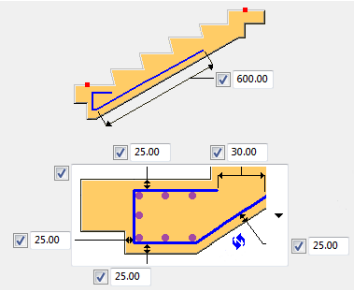
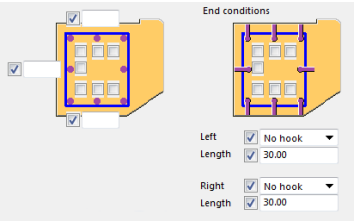
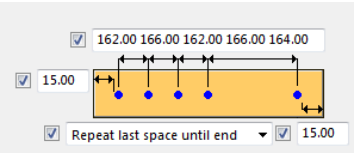
Option	Description
	Bar B simple top ending.
	Bar B top ending bent once to fit the top geometry of the stairs.
	Bar B top ending bent twice to fit the top geometry of the stairs. Last leg length is defined by the cover thickness.
	Bar B top ending bent twice to fit the top geometry of the stairs.

Reinforcing bar spacing

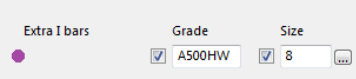
Option	Description
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

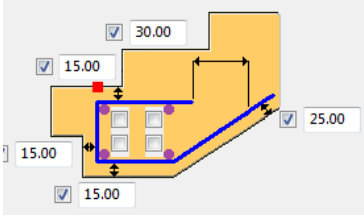
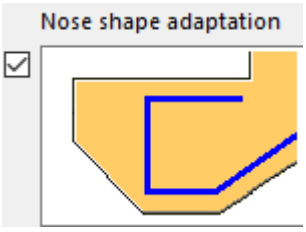
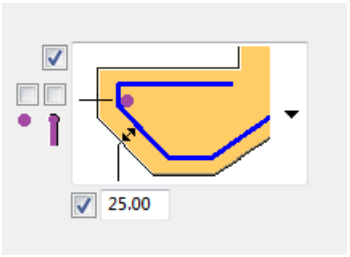
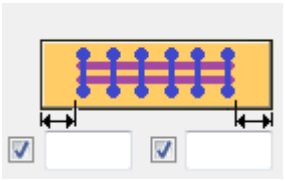
Bar C tab

Use the **Bar C** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group.

Option	Description
	Geometry and concrete cover thickness.
	Define which extra side bars are created, their end conditions, and the concrete cover.
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

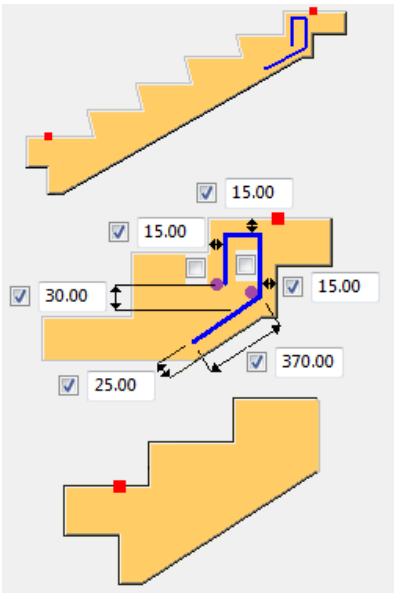
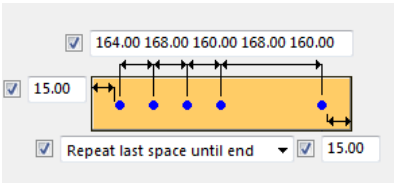
Extra I bars

Option	Description
	The purple color in the dialog box represents the extra I bars.

Option	Description
	Define whether to create the extra I bars. Select the check boxes next to the purple points.
 	Select the nose shape adaptation. The first check box (the dot on the left) creates the extra I bar and the second creates a hook on it.
	Cover thickness of the reinforcing bars.

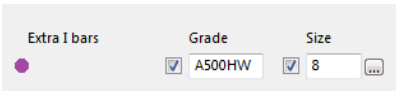
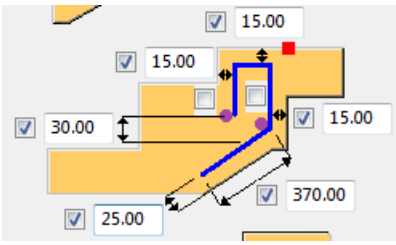
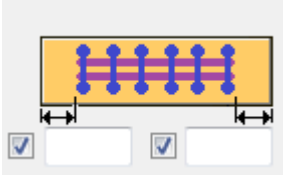
Bar D tab

Use the **Bar D** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group.

Option	Description
	Geometry and concrete cover thickness.
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

Extra I bars

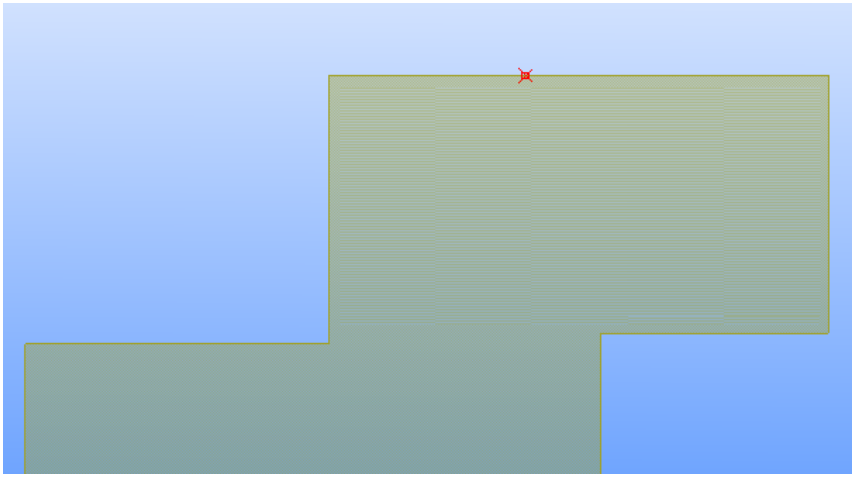
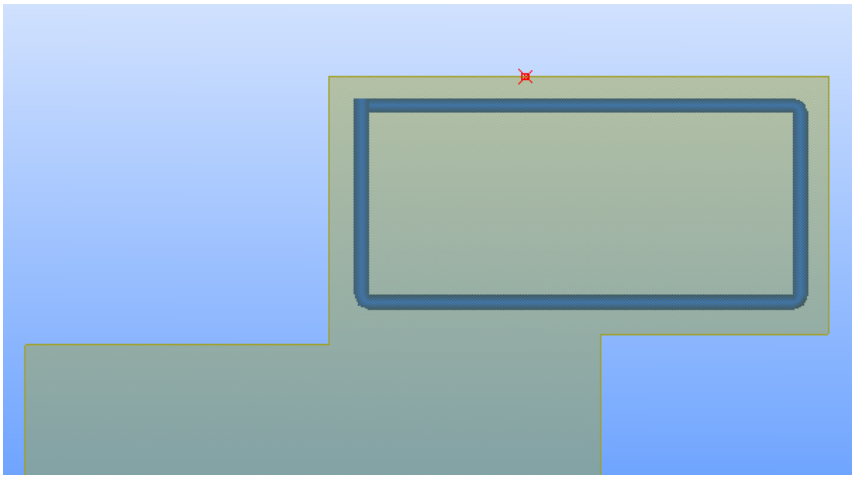
Extra I bars are straight bars crossing bar D. You can create up to four extra I bars.

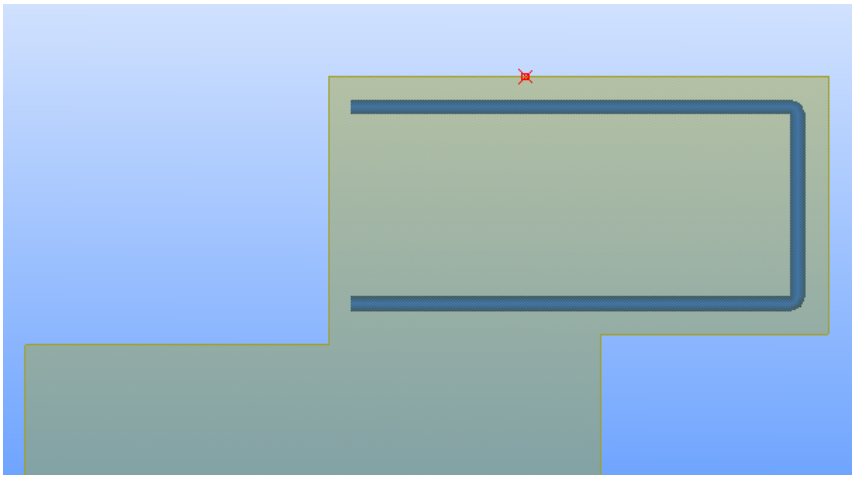
Option	Description
	The purple color in the dialog box represents the extra I bars.
	Define whether to create the extra I bars. Select the check boxes next to the purple points.
	Cover thickness of the reinforcing bars.

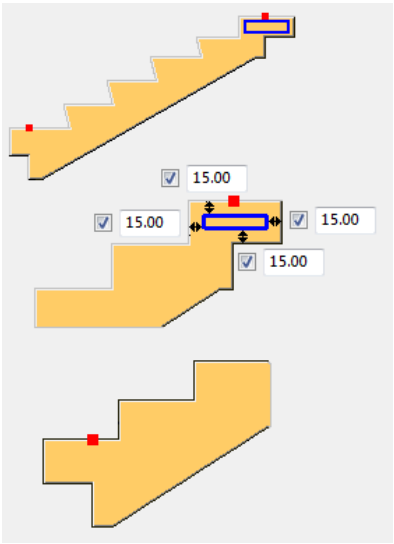
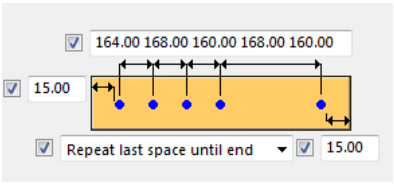
Bar E tab

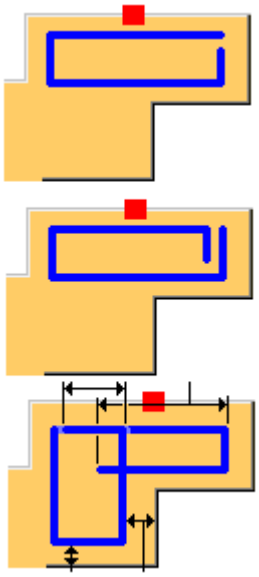
Use the Bar E tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group.

Create options

Option	Description
No	Bar is not created. 
Stirrup	Bar is created as stirrup. 

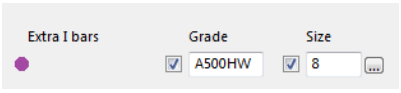
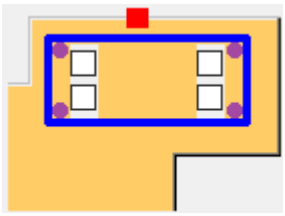
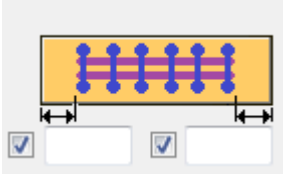
Option	Description
Pin	Bar is created as pin. 

Option	Description
	Geometry and concrete cover thickness.
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

Option	Description
	Shape of the stirrups.
Parallel with step slope	**Yes** Create parallel with sloped step shape.
	No Create as a rectangle.

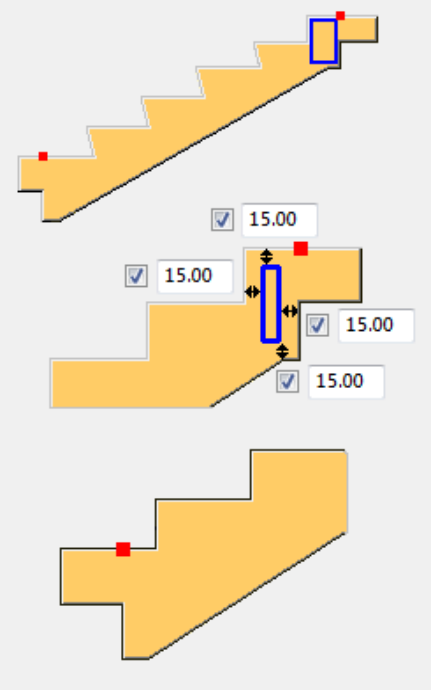
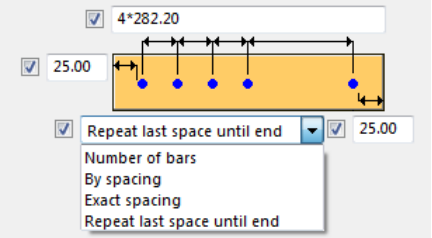
Extra I bars

Extra I bars are straight bars crossing bar D. You can create up to four extra I bars.

Option	Description
	The purple color in the dialog box represents the extra I bars.
	Define whether to create the extra I bars. Select the check boxes next to the purple points.
	Cover thickness of the reinforcing bars.

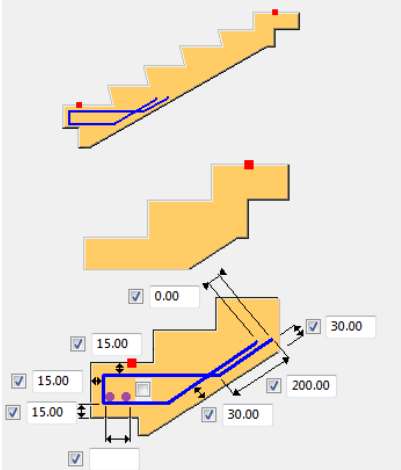
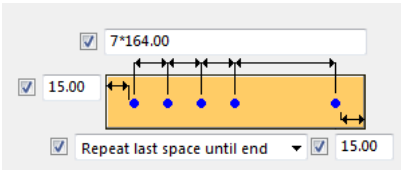
Bar F tab

Use the **Bar F** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group.

Option	Description
	Geometry and concrete cover thickness.
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

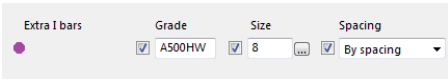
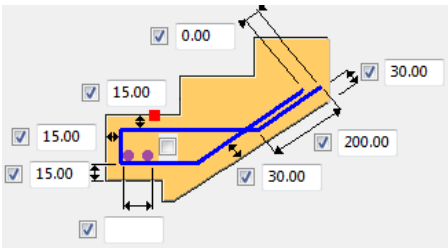
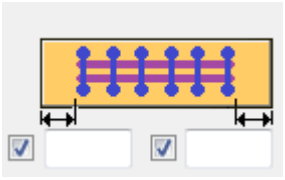
Bar G tab

Use the **Bar G** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group.

Option	Description
	Geometry and concrete cover thickness.
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

Extra I bars

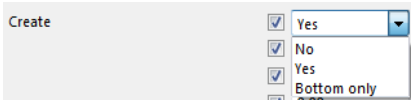
Extra I bars are straight bars crossing bar G. You can create up to four extra I bars.

Option	Description
	The purple color in the dialog box represents the extra I bars.
	Define whether to create extra I bars. Select the check boxes next to the purple points.
	Cover thickness of the reinforcing bars.

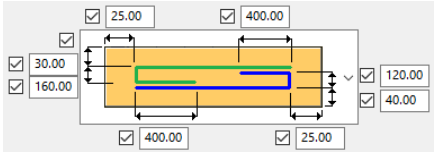
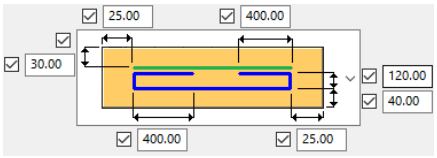
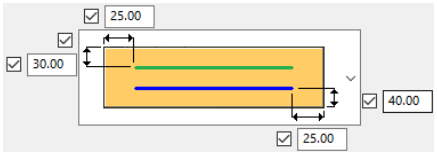
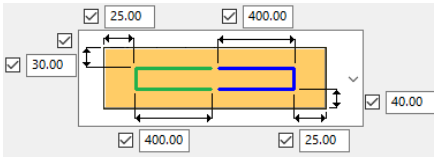
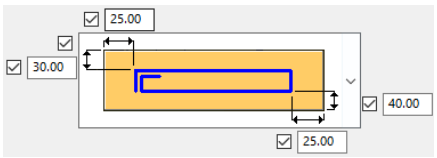
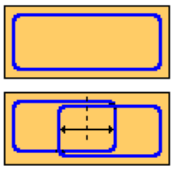
Bar H tab

Use the **Bar H** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group.

Create option

Option	Description
	• Do not create bar H • Create bar H • Create only bottom bar H reinforcing bar group

Bar H geometry option

Option	Description
	Two bent reinforcing bar groups. Define geometry and concrete cover thickness.
	Top reinforcing bar group is straight, bottom reinforcing bar group is bent on both sides. Define geometry and concrete cover thickness.
	Two straight reinforcing bar groups. Define geometry and concrete cover thickness.
	Two reinforcing bar groups. Define geometry and concrete cover thickness.
	Reinforcing bar group that is bent on one side. Define the concrete cover thickness and whether to create hooks. You can create multiple overlapping stirrups in a row. Define the overlap dimension and the number of stirrups. 

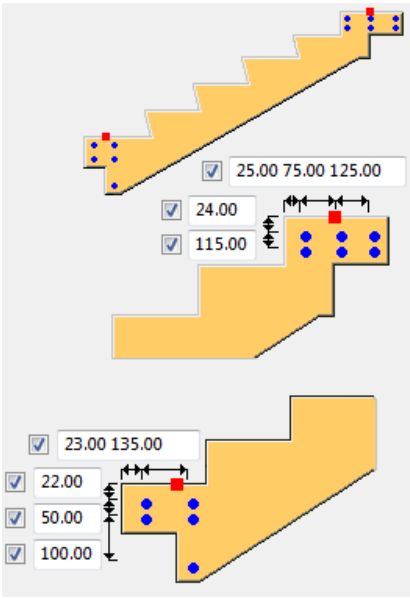
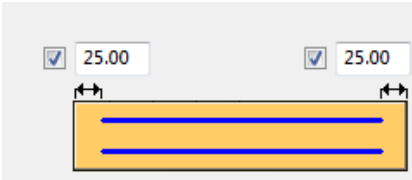
Properties

You can define separate properties for bottom and top rebars.

Option	Description
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement.
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.

Bar I tab

Use the **Bar I** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group.

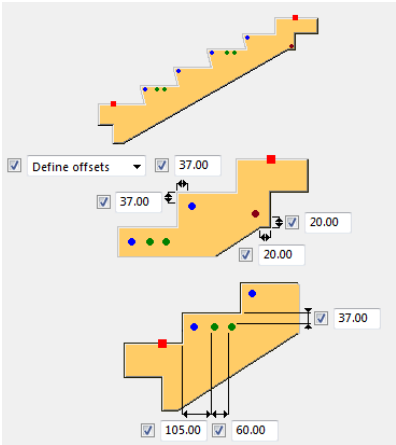
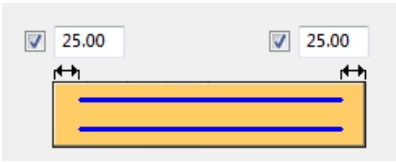
Option	Description
	Positioning, spacing and concrete cover thickness.
	Concrete cover thickness of the reinforcing bar groups.

Bar J tab

Use the **Bar J** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group. You can create several bar J reinforcing bar groups. Each group has its own color representation on the **Bar J** tab.

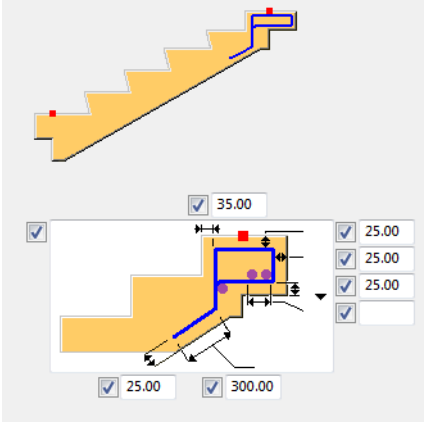
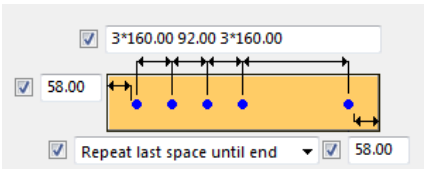
Color group options

Option	Description
	Use the color group option to define which combination of bar J reinforcing bar groups (blue, green, brown) is created.

Option	Description
	Positioning and concrete cover thickness for reinforcing bar groups based on the color representation.
	Concrete cover thickness for all reinforcing bar groups.

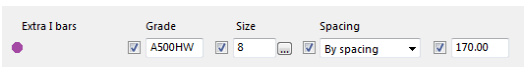
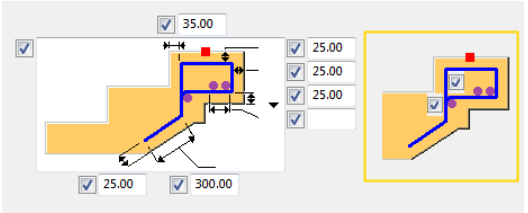
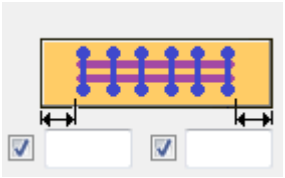
Bar K tab

Use the **Bar K** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group.

Option	Description
	Select the geometry and define the concrete cover thickness.
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

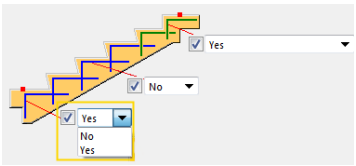
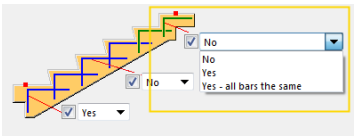
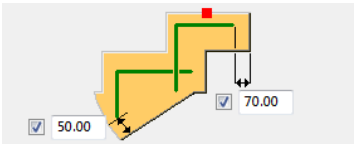
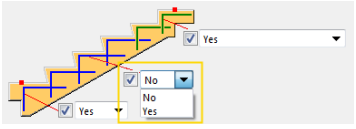
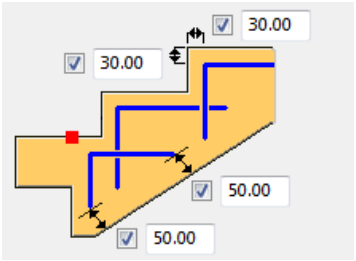
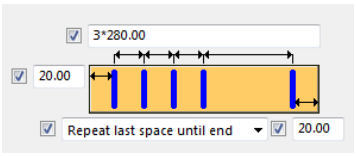
Extra I bars

Extra I bars are straight bars crossing bar K. You can create up to three extra I bars.

Option	Description
	The purple color in the dialog box represents the extra I bars.
	Define whether to create extra I bars by selecting the check boxes next to the purple points.
	Cover thickness of the reinforcing bars.

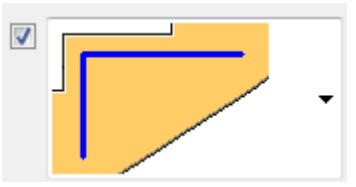
Bar L tab

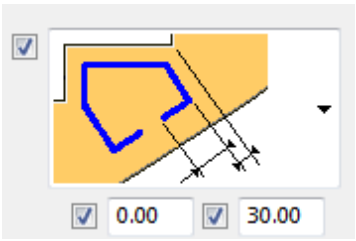
Use the **Bar L** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group.

Option	Description
	Define whether an L shaped reinforcing bar group is created at the stair footing.
	Define whether an L shaped reinforcing bar group is created at the stair top or whether all bars are created with the same geometry.
	Define offsets for the reinforcing bar groups in the two top steps.
	Define whether reinforcing bars are created between the top and bottom step.
	Define geometry for reinforcing bar groups in all steps, except for the two top steps.
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

Bar L geometry options

The geometry options affect the middle and the bottom steps.

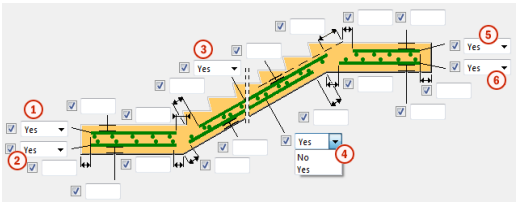
Option	Description
	L shape reinforcing bar groups.

Option	Description
	L shape reinforcing bar groups with hooks.

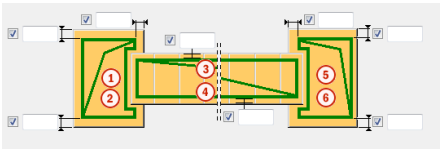
Mesh picture tab

Use the **Mesh picture** tab to define bottom landing meshes, stair meshes and top landing meshes.

Side view

		Description
	**1**	Bottom landing mesh (top). Define offsets and cover thickness of the mesh.
	2	Bottom landing mesh (bottom). Define offsets and cover thickness of the mesh.
	3	Stair mesh (top). Define offsets and cover thickness of the mesh.
	4	Stair mesh (bottom). Define offsets and cover thickness of the mesh.
	5	Top landing mesh (top). Define offsets and cover thickness of the mesh.
	6	Top landing mesh (bottom). Define offsets and cover thickness of the mesh.

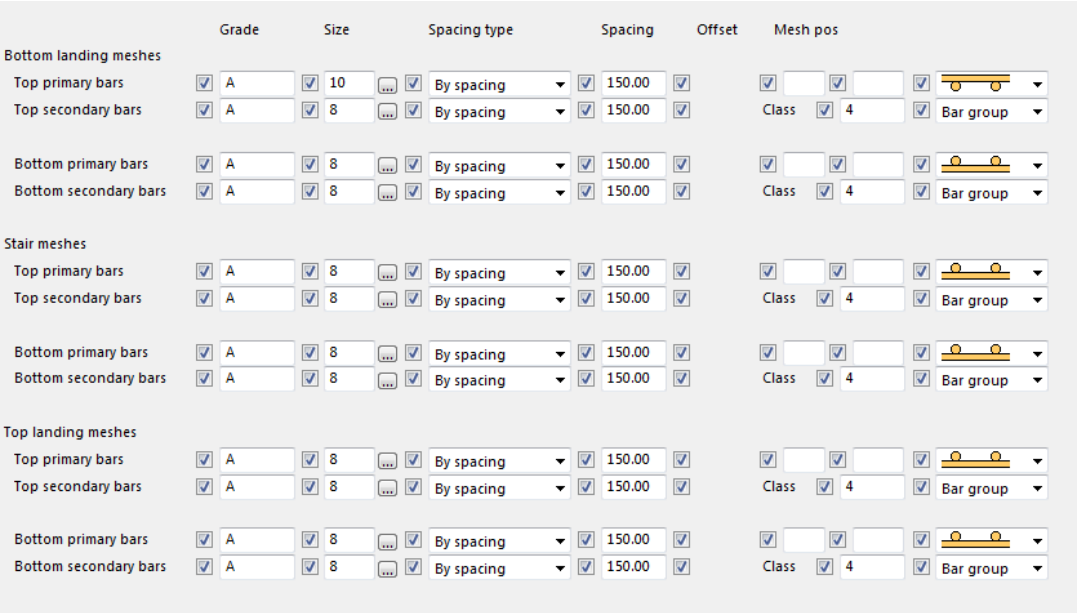
Top view

		Description
	**1**	Cover thickness of bottom landing mesh (top).
	2	Cover thickness of bottom landing mesh (bottom).
	3	Cover thickness of stair mesh (top).
	4	Cover thickness of stair mesh (bottom).

		Description
	5	Cover thickness of top landing mesh (top).
	6	Cover thickness of top landing mesh (bottom).

Mesh attributes tab

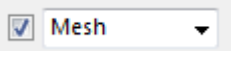
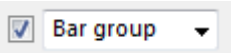
Use the **Mesh attributes** tab with the **Mesh picture** tab to define the properties, positioning and creation type of bottom landing meshes, stair meshes and top landing meshes.



Cross bar location

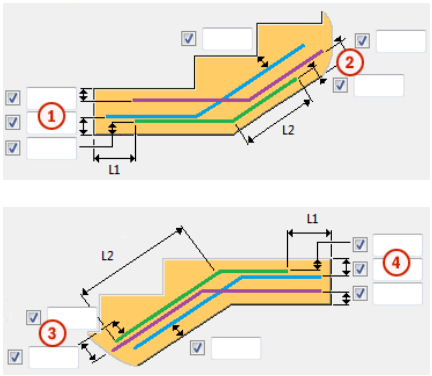
	Description
	Cross bar above.
	Cross bar below.

Creation type

	Description
	Create mesh as reinforcement mesh.
	Create mesh as two independent reinforcing groups.

Mesh bars tab

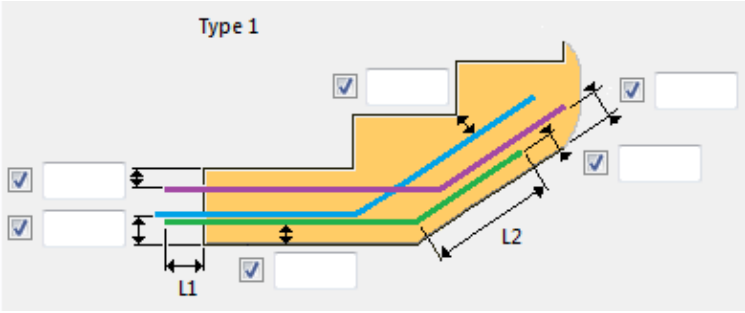
Use the **Mesh bars** tab to define the geometry, concrete cover thickness, reinforcing bar spacing, and reinforcing bar properties of the mesh bars in the top and bottom landing. You can create up to three reinforcing bar groups.

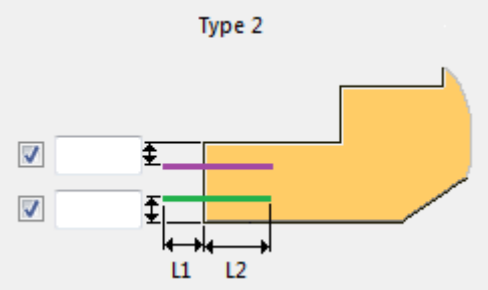
		Description
	**1**	Cover thicknesses of all mesh bar groups in the stair bottom landing. Use the color representation in the dialog box.
	2	Vertical cover thicknesses of all mesh bar groups in the stair bottom landing. Use the color representation in the dialog box.
	3	Cover thicknesses of all mesh bar groups in the stair top landing. Use the color representation in the dialog box.
	4	Vertical cover thicknesses of all mesh bar groups in the stair top landing. Use the color representation in the dialog box.

Bottom anchor bars tab

Use the **Bottom anchor bars** tab to define the geometry, concrete cover thickness, reinforcing bar spacing, and reinforcing bar properties of the bottom anchor bars. You can create up to three reinforcing bar groups.

Creation type

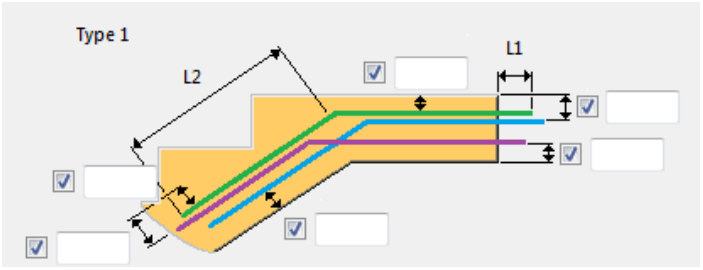
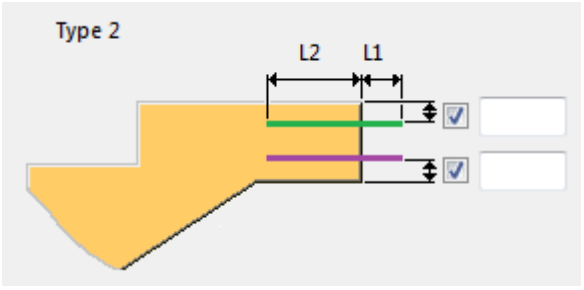
Option	Description
No	No reinforcing bar group created.
Type1	L shape anchor reinforcing bar group. 

Option	Description
Type2	Simple straight anchor reinforcing bar group. 

Top anchor bars tab

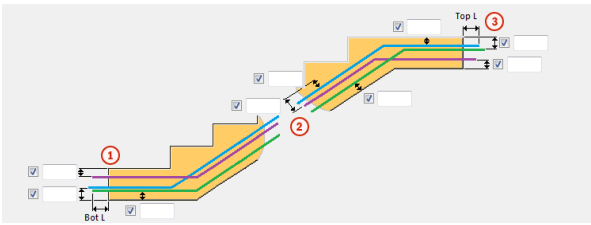
Use the **Top anchor bars** tab to define the geometry, concrete cover thickness, reinforcing bar spacing, and reinforcing bar properties of the top anchor bars. You can create up to three reinforcing bar groups.

Creation type

Option	Description
No	No reinforcing bar group created.
Type1	L shape anchor reinforcing bar group. 
Type2	Simple straight anchor reinforcing bar group. 

Z anchor bars tab

Use the **Z anchor bars** tab to define the geometry, concrete cover thickness, reinforcing bar spacing, and reinforcing bar properties of the Z anchor bars. You can create up to three reinforcing bar groups.

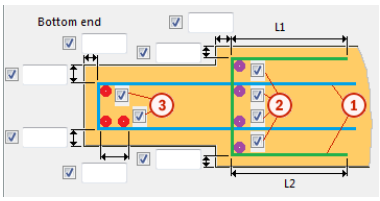
		Description
	**1**	Cover thicknesses of all Z anchor bar groups in the stair bottom landing. Use the color representation in the dialog box.
	2	Vertical cover thicknesses of all Z anchor bar groups in the stair main part. Use the color representation in the dialog box.
	3	Cover thicknesses of all Z anchor bar groups in the stair top landing. Use the color representation in the dialog box.
	Bot L	Length of the Z anchor bars that extend from the stair bottom landing.
	Top L	Length of the Z anchor bars that extend from the stair top landing.

Landing end bars tab

Use the **Landing end bars** tab to reinforce the stair landings and to define the geometry, concrete cover thickness, reinforcing bar spacing, and bar properties of the landing end bars.

Bottom end

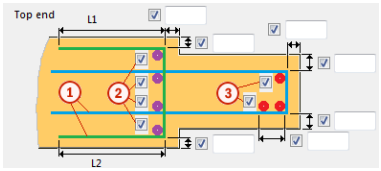
Reinforcement of bottom landing.

		Description
	**1**	Landing end bars. U-shape geometry follows the shape of the landing based on the defined cover thicknesses. Color representation in the picture: blue, green.
	2	Define whether to create extra cross bars. Select the check boxes next to the purple points.

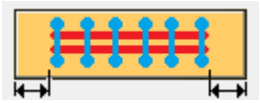
		Description
	3	Define whether to create extra cross bars. Select the check boxes next to the red points.

Top end

Reinforcement of top landing.

		Description
	**1**	Landing end bars. U-shape geometry follows the shape of the landing based on defined cover thicknesses. Color representation in the picture: blue, green.
	2	Define whether to create extra cross bars. Select the check boxes next to purple points.
	3	Define whether to create extra cross bars. Select the check boxes next to red points.

Side cover thickness

Option	Description
	Side cover thickness of all reinforcing bar groups defined on the **Landing end bars** tab.

UDA tab

Use the **UDA** tab to define user-defined attributes for the stairs. You can define multiple UDAs. UDAs can be used as filters, and they can be displayed in drawings and reports.

Configuration tab

Use the **Configuration** tab to define the default cover thicknesses and bending radiuses of all reinforcing bars created by **Reinforced Concrete Stair (95)**.

Cover thickness for reinforcing bar groups A - C

Define the default cover thickness for reinforcing bar groups A - C.

Option	Description
Value	Default cover thickness defined by a value.
Rebar diameter	Default cover thickness multiplied by reinforcing bar diameter.

General cover thickness bars D - L

Define the default cover thickness for reinforcing bar groups D - L and all reinforcing bars except the reinforcing bar groups A - C and meshes.

Option	Description
Value	Default cover thickness defined by a value.
Rebar diameter	Default cover thickness multiplied by reinforcing bar diameter.

Bending radius

Define the default bending radius of all reinforcing bars.

Option	Description
Rebar_database.inp	Default bending radius defined by `rebar_database.inp` file.
Relative to diameter	Default bending radius according to the reinforcing bar diameter.
Bending radius	Default bending radius defined by value.

Mesh cover thickness

Define the default cover thickness for the meshes created by **Reinforced Concrete Stair (95)**.

Foundations

This section introduces components that can be used in concrete foundations.

Click the links below to find out more:

- Precast foundation block (1028) (page 3811)
- Concrete foundation (1030) (page 3818)

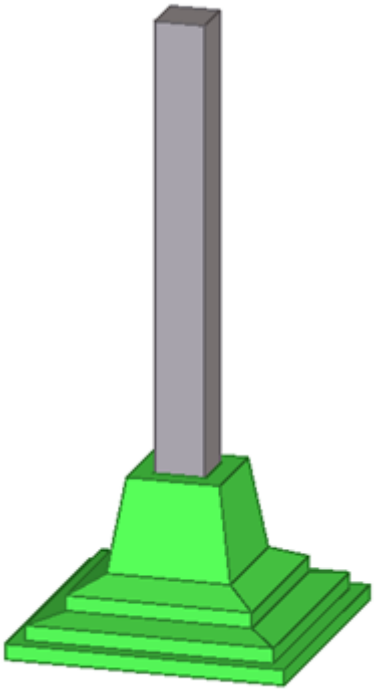
Precast foundation block (1028)

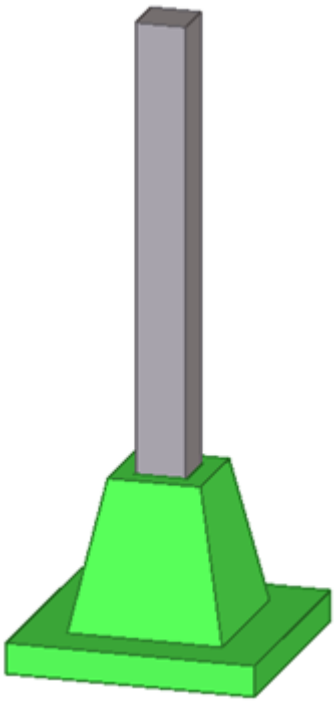
Precast foundation block (1028) creates a concrete foundation block. The block can be divided in three sections. All three sections can be dimensioned separately.

Parts created

- Foundation block
- Embed
- Regulator

Use for

Situation	Description
	Precast foundation block with three sections.

Situation	Description
	Precast foundation block with one section.

Selection order

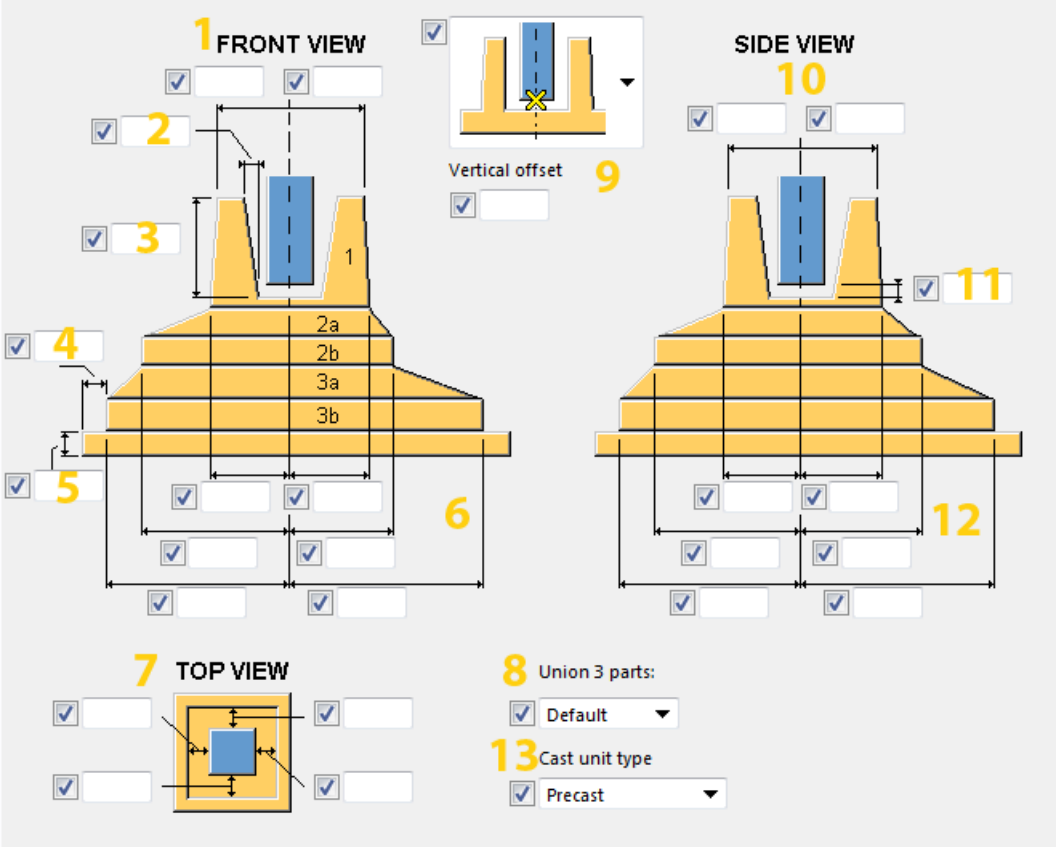
1. Select the main part (concrete column).
2. Pick a position.

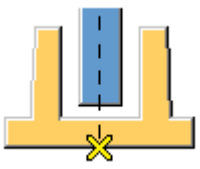
   The detail is created automatically.

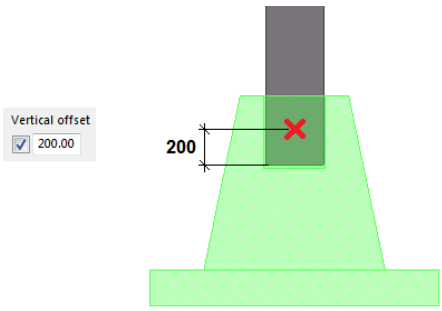
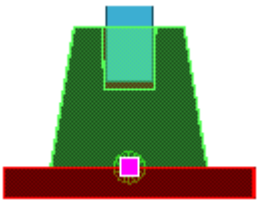
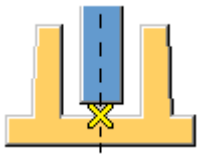
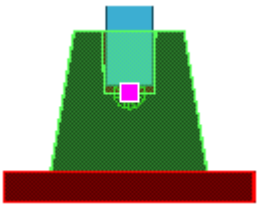
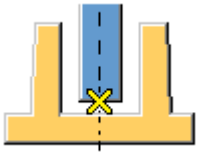
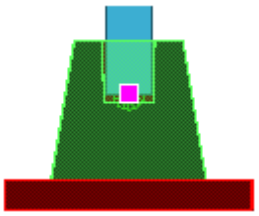
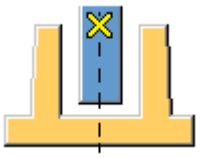
Picture tab

Use the **Picture** tab to dimension the foundation block.

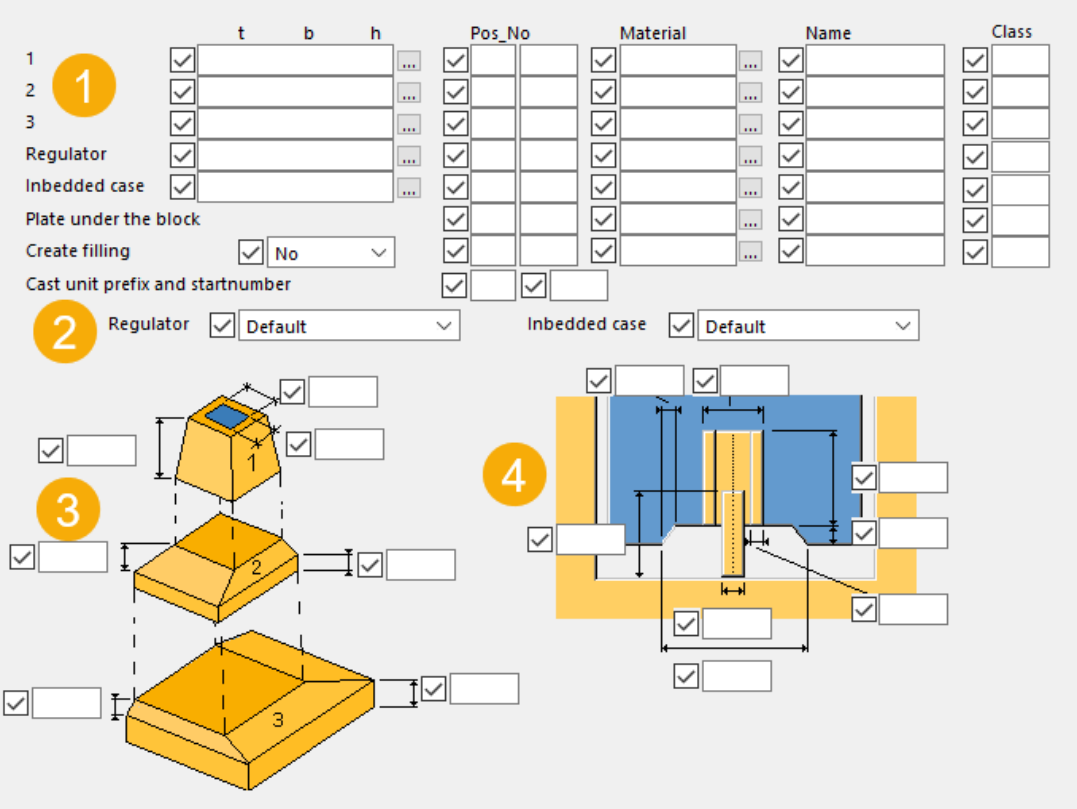
The foundation block can be dimensioned in both the front view and in the side view.



	Description		
1	Width of the top chord in section 1 (in front view).		
2	Width of the inclined column cut-out.		
3	Height of the column cut-out.		
4	Offset of the plate that is under the block. The offset is relative to the foundation block.		
5	Thickness of the plate under the block.		
6	Width of the bottom chord in section 1 (in front view). Width of section 2 and section 3 (in front view).		
7	Distance between the foundation block and the sides of the column.		
8	Select how the sections in the foundation block are connected.		
9	Select the reference point of the foundation block.		Component insertion point is at the bottom side of the foundation block.

<table>
<tr><th></th><th colspan="3">Description</th></tr>
<tr><td></td><td>Additionally, you can set the vertical offset:
 
</td><td></td><td>Note that the red part (additional plate under the block) is not taken into account.
  </td></tr>
<tr><td></td><td></td><td>  </td><td>Component insertion point is at the bottom side of the cut-out of the column.
  </td></tr>
<tr><td></td><td></td><td>  </td><td>Component insertion point is at the bottom side of the column.
  </td></tr>
<tr><td></td><td></td><td>  </td><td>Component insertion point is at the top of the foundation block.</td></tr>
<tr><td>10</td><td colspan="3">Width of the top chord in section 1 (in side view).</td></tr>
<tr><td>11</td><td colspan="3">Distance from the bottom of the column to the foundation block.</td></tr>
<tr><td>12</td><td colspan="3">Width of the bottom chord in section 1 (in side view).
Width of section 2 and section 3 (in side view).</td></tr>
<tr><td>13</td><td colspan="3">Select the cast unit type.</td></tr>
</table>

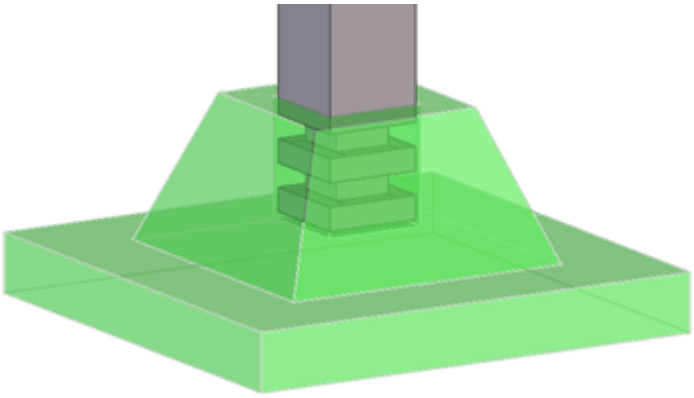
Parts tab
Use the **Parts** tab to define the profiles for the foundation block, embed and regulator, and the embed dimensions.

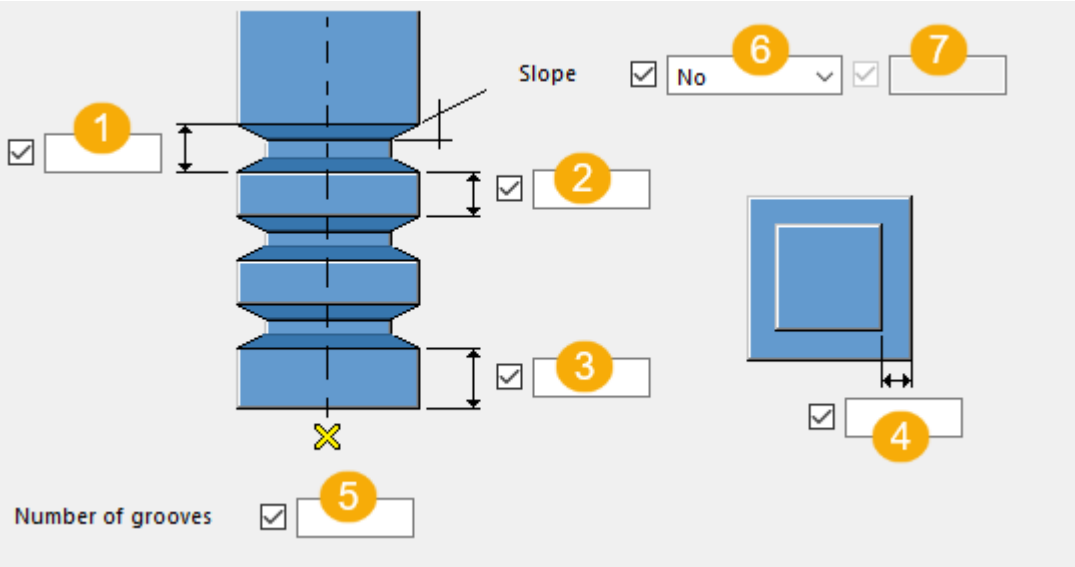


Option	Description
1	Foundation block profiles, regulator and embed properties. If you do not select any material, **Precast foundation block (1028)** uses the same material as the column. Select **Yes** in **Create filling** to create a filling between the column and the first created block.
2	Select whether the regulator and the embed is created, and how they are connected to the foundation block.
3	Dimensions of the three sections of the foundation block. Additionally, you can define the column cut-out in the upper section. For sections 2 and 3 you can define the height of the inclined plane.
4	Dimensions of the embed.

Grooves tab
Use the **Grooves** tab to define the grooves.

Example:





	Description
1	Height of the groove.
2	Distance between the grooves.
3	First distance related to the bottom of the column.
4	Depth for the grooves.
5	Number for the grooves.
6	Select whether to define a slope angle or dimension for the grooves.
7	Define the slope angle or dimension value.

General tab

Click the link below to find out more:

Analysis tab

Click the link below to find out more:

Analysis tab

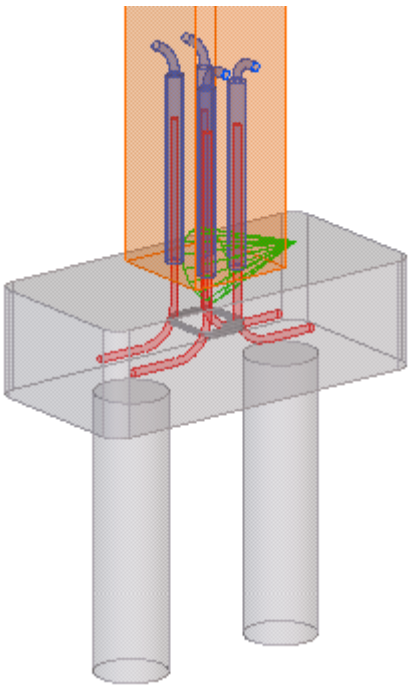
Concrete foundation (1030)

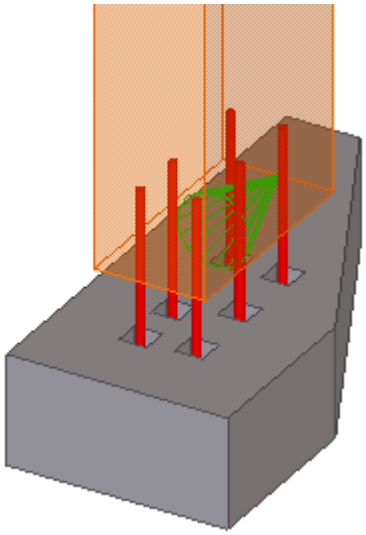
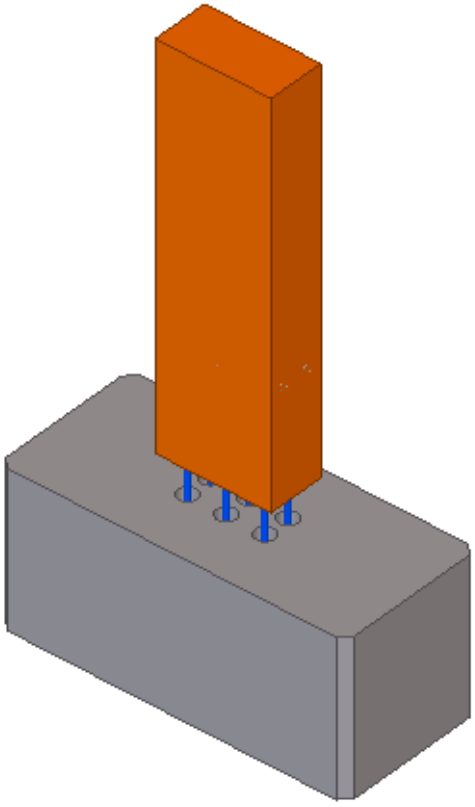
Concrete foundation (1030) creates a concrete foundation plate at the bottom of a selected concrete column.

Objects created

- Concrete foundation plate
- Injection tubes and injection hoses in concrete column
- Up to 4 concrete piles under the foundation plate (optional)
- Stirrups for the reinforcing bars

Use for

Situation	Description
	Concrete foundation plate with chamfers, piles, injection tubes with curved injection hoses, reinforcing bars and stirrups.

Situation	Description
	Concrete foundation plate with more than 4 edges, recesses in the foundation plate, and multiple reinforcing bars.
	Concrete foundation plate with multiple reinforcing bars.

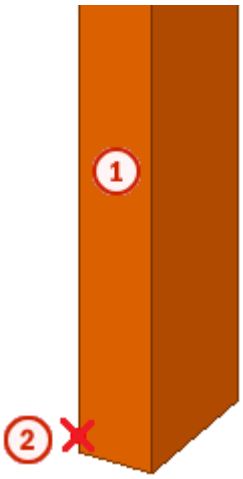
Selection order

1. Select a concrete column.

2. Pick a point.

   The concrete foundation plate is created automatically when the point is picked.

Part identification key

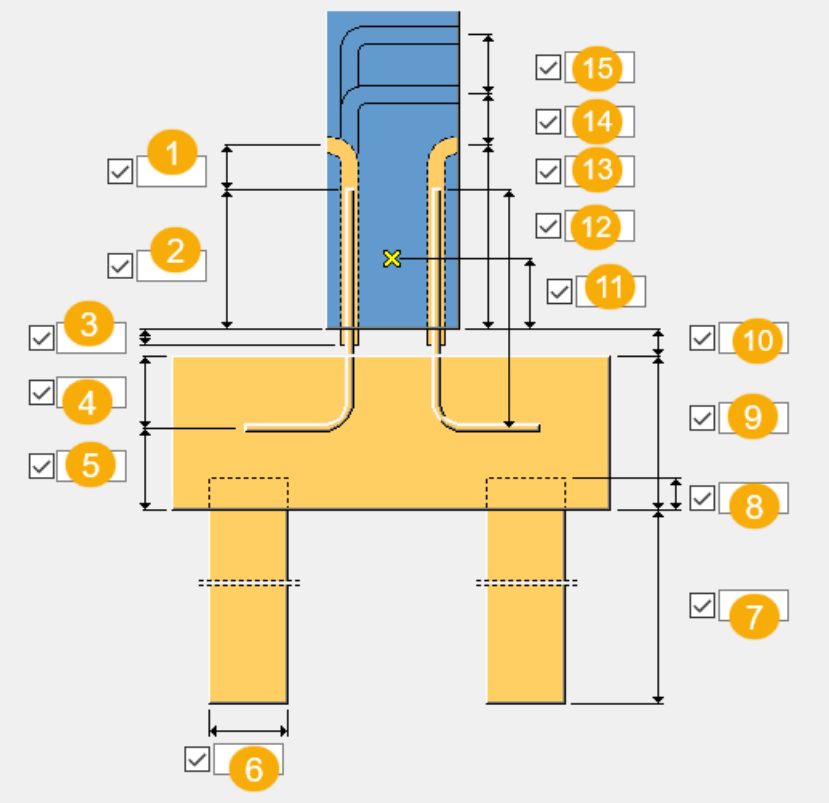


	Part
1	Concrete column
2	Point The point defines the new bottom level of the column.

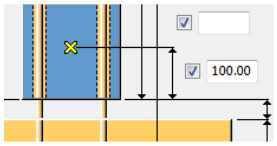
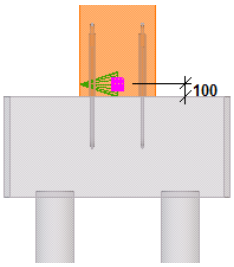
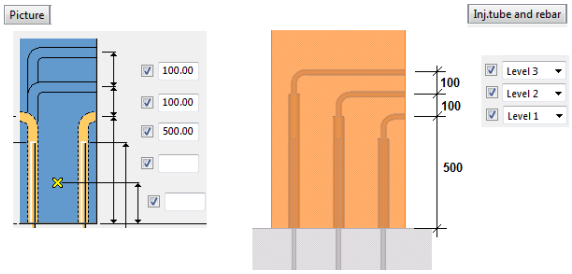
Picture tab

Use the **Picture** tab to control the dimensions of the reinforcing bars and injection tubes.

Dimensions

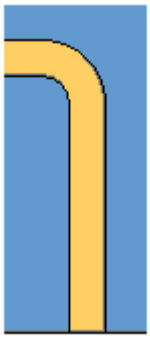
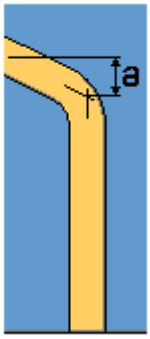
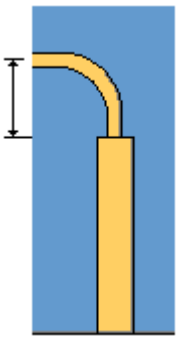
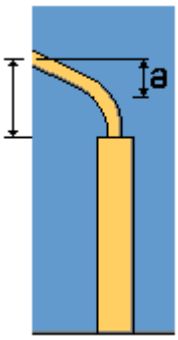


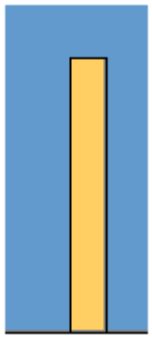
	Description	Default
1	Length of the injection tube extension from the top of the reinforcing bar.	`180 mm`
2	Reinforcing bar height in the column.	`400 mm`
3	Offset of the injection tube from the bottom of the column.	
4 **5**	Reinforcing bar top and bottom cover thickness. The bottom cover thickness is used only if you do not define the top cover thickness.	`0,5 * foundation plate thickness`
6	Pile thickness. The pile thickness value on the **Parts** tab overrides this value.	`300 mm`
7	Distance from the bottom of the foundation plate to the bottom of the pile.	`5000 mm`
8	Pile depth in the foundation plate.	`50 mm`
9	Foundation plate thickness.	`800 mm`

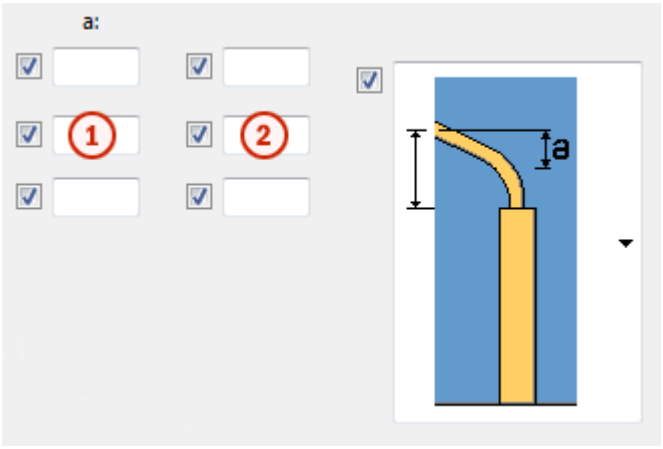
	Description	Default
10	Distance between the column and the foundation plate.	0 mm
11	Vertical offset from the picked point.  	0 mm
12	Reinforcing bar height.	800 mm 400 mm + 50% of height of concrete footing
13	Injection tube elevation 1.	500 mm
14, **15**	Injection tube elevations 2 and 3. Vertical offset for the injection tubes if the tubes are pointing to the same direction. Define the level to use on the **Inj. Tube and rebar** tab. 	

Injection tube

Select the shape and dimensions of the injection tubes. Sloping injection hoses are fitted to the column edge.

Option	Description
	Injection tube in one piece, 90 degree angle.
	Injection tube in one piece. Use value **a** to define the slope of the tube.
	Injection tube and a 90 degree injection hose.
	Injection tube and a sloped injection hose. Use value **a** to define the slope of the hose.

Option	Description
	Straight injection tube. No injection tube opening is created.

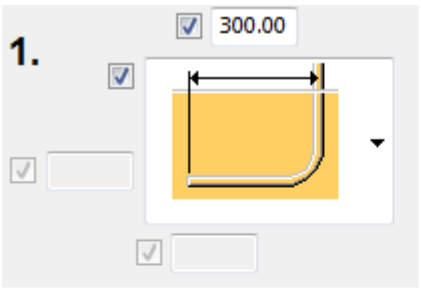


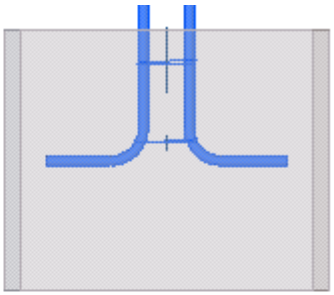
	Description	Default
1	Define the height of a sloped injection tube or hose angle.	`30 mm`
2	Define the height of a curved injection hose.	`0 mm`

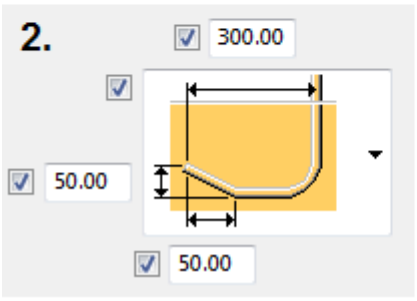
Reinforcing bar hook length

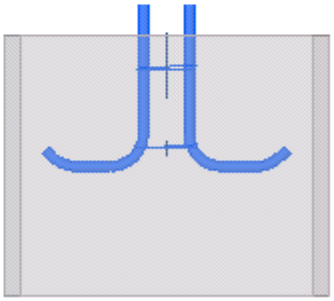
Define the reinforcing bar hook length. You can define also the end angle of the reinforcing bar.

The default value is `10 * reinforcing bar diameter`.









Parts tab

Use the **Parts** tab to control the profile properties of the foundation plate, foundation piles, stirrups and injection tubes.

Part properties

Define the part profile properties.

If you leave the pile thickness value empty on the **Parts** tab, the pile thickness value defined on the **Picture** tab is used instead together with the **Pile profile type prefix (e.g. D)**.

Option	Description
t, **b**, **h**	Define the thickness, width, and height of the foundation plate and piles, or select the profile from the profile catalog.
Pos_No	Prefix and a start number for the part position number.
Material	Material grade.
Name	Name that is shown in drawings and reports.
Class	Part class number.
Comment	Add a comment for the part.

Reinforcing bars

Option	Description
Type of reinforcing bars	Select the profile type for the reinforcing bars. • **Default**: the same as **Reinforcing bar** • **Poly-profile**: profile catalog • **Reinforcing bar**: reinforcing bar catalog • **Polyprofile (as sub-assembly)**: profile catalog. Create the polyprofile as a sub-assembly to the foundation part.
Reinforcement bars	Size of the reinforcing bars.
Radius	Radius of the reinforcing bar hook.
Reinf. bars belong to	Define to which part the reinforcing bars belong. • **Default**: the same as **Columns** • **Concrete massive**: reinforcing bars belong to the concrete foundation plate. • **Column**: reinforcing bars belong to the column. • **Loose part**: reinforcing bars are not connected to any part. • **Neighbor**: reinforcing bars belong to the neighbor part. Use the name or class to find the part.

Injection tubes and curve

Option	Description
Injection tubes **Curve**	Size of the injection tubes. Size of the injection hoses.
Article number	Article number for the injection hose and the injection tube. The entered values are saved to the user-defined attribute of the part.
	Define how the injection hoses are connected to the injection tubes. • **Default**: injection hoses are loose parts. • **Part-add gain**: injection hoses are added to the injection tubes.

Option	Description
	• **Weld gain**: injection hoses are welded to the injection tubes. • **Cast unit column**: injection hoses are added to the column. • **Weld to column**: injection hoses are welded to the column.

Option	Description
Pile profile type prefix (e.g. D)	Default parametric profile prefix for the piles. This value works only if you have set the pile thickness on the **Picture** tab. You can override this value by setting the pile thickness on the **Parts** tab.
Reinf. bar profile type prefix (e.g.D)	Default parametric profile prefix for the reinforcing bar profiles.

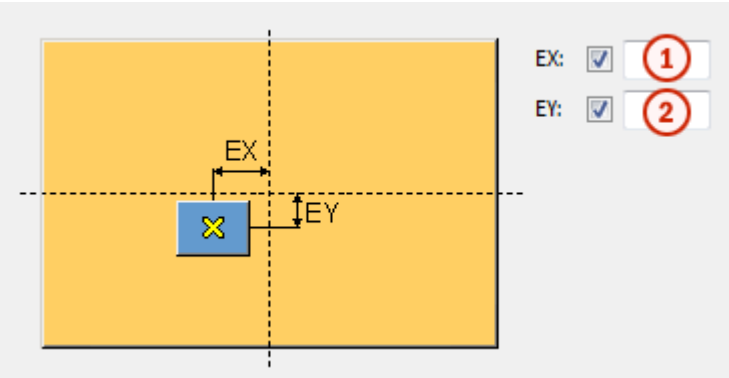
Massive tab

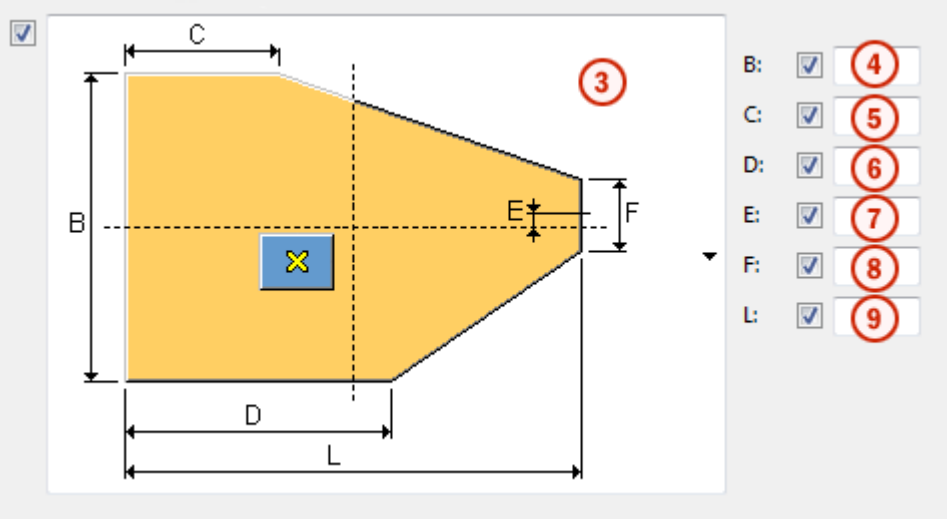
Use the **Massive** tab to control the shape and the dimensions of the concrete foundation plate and the recesses.

Concrete foundation plate type

Option	Description
Type of massive	Select the plate type for the rectangular foundation plate.
Massive direction	Select the direction of the foundation plate.

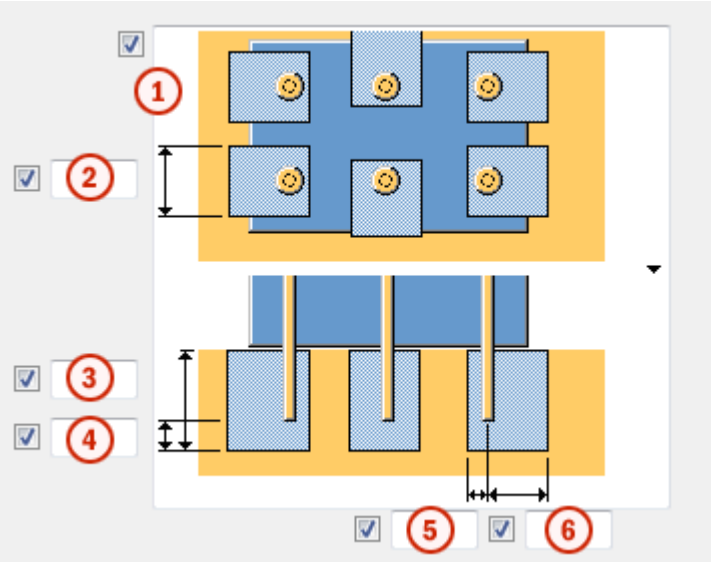
Concrete foundation plate dimensions





	Description
1	Offset from the concrete column in the x direction.
2	Offset from the concrete column in the y direction.
3	Select the shape of the foundation plate.
4	Width of the foundation plate.
5 **6**	Length of the straight part of the foundation plate.
7	Offset from the foundation plate center line.
8	Width of the straight part of the foundation plate.
9	Length of the foundation plate.

Recess dimensions



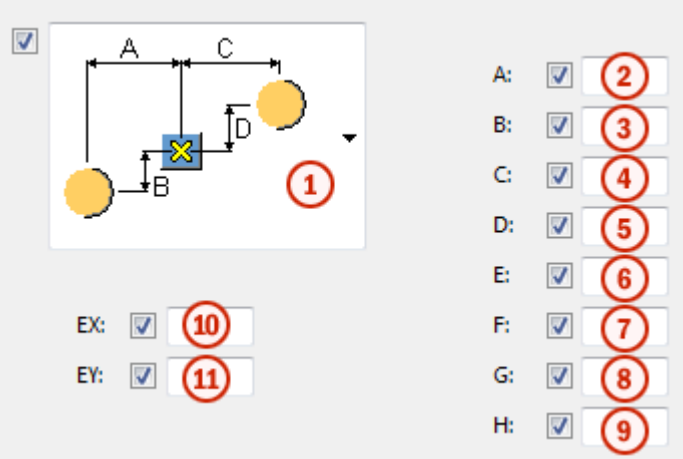
	Description
1	Select the type of the recess.
2	Width of the recesses.
3	Depth of the recesses.
4	Distance between the bottom of the reinforcing bar and the bottom of the recess.
5	Distance between the center line of the reinforcing bar and the left side of the recess.
6	Distance between the center line of the reinforcing bar and the right side of the recess.

Option	Description
Chamfer type for rectangular massive	Chamfer type for the rectangular foundation plate.
Chamfer X	Chamfer size in the x direction.
Chamfer Y	Chamfer size in the y direction.

Piles tab

Use the **Piles** tab to control the position of the foundation piles.

Pile positions



	Description
1	Select the position and the offset of the piles.
2	Distance between the first pile and the column in the x direction.
4	Distance between the second pile and the column in the x direction.
6	Distance between the third pile and the column in the x direction.
8	

	Description
	Distance between the fourth pile and the column in the x direction.
3	Distance between the first pile and the column in y direction.
5	Distance between the second pile and the column in y direction.
7	Distance between the third pile and the column in y direction.
9	Distance between the fourth pile and the column in y direction.
10	Pile offset from the concrete column in the x direction.
11	Pile offset from the concrete column in the y direction.

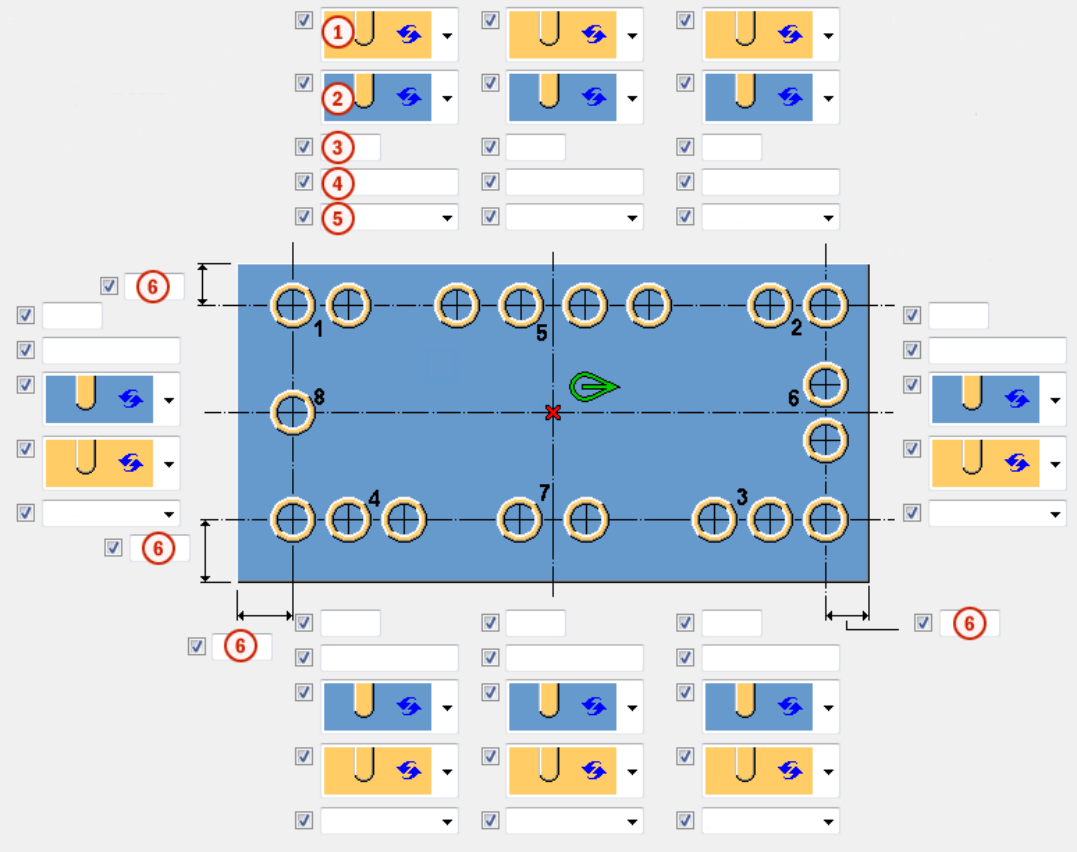
Option	Description
Piles direction	Define the direction of the piles.
Join piles to massive	Define how the piles are connected to the foundation plate.

Inj. tube and rebar tab

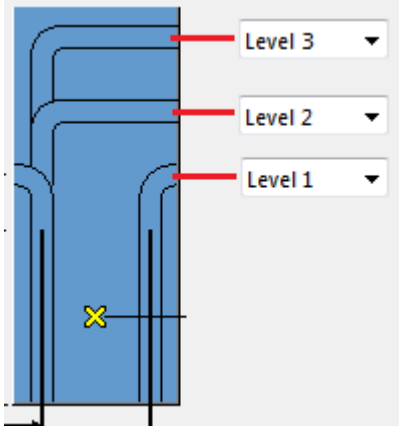
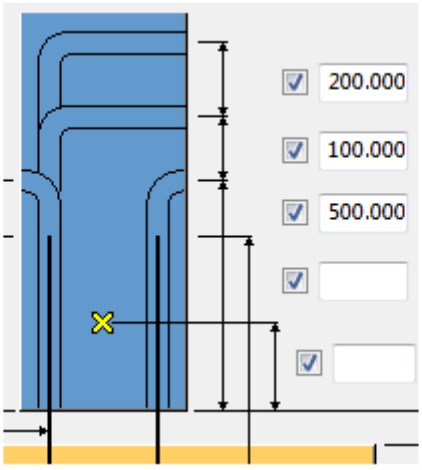
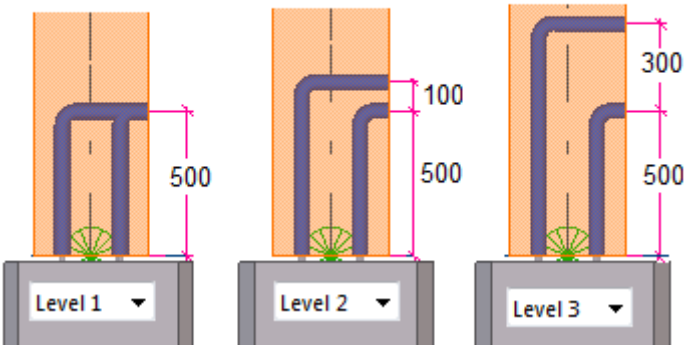
Use the **Inj. tube and rebar** tab to control the creation, number and position of the reinforcing bars and injection tubes.

Option	Description
Create	Select which parts are created.
Injection tubes bend radius	Radius of the injection tubes.

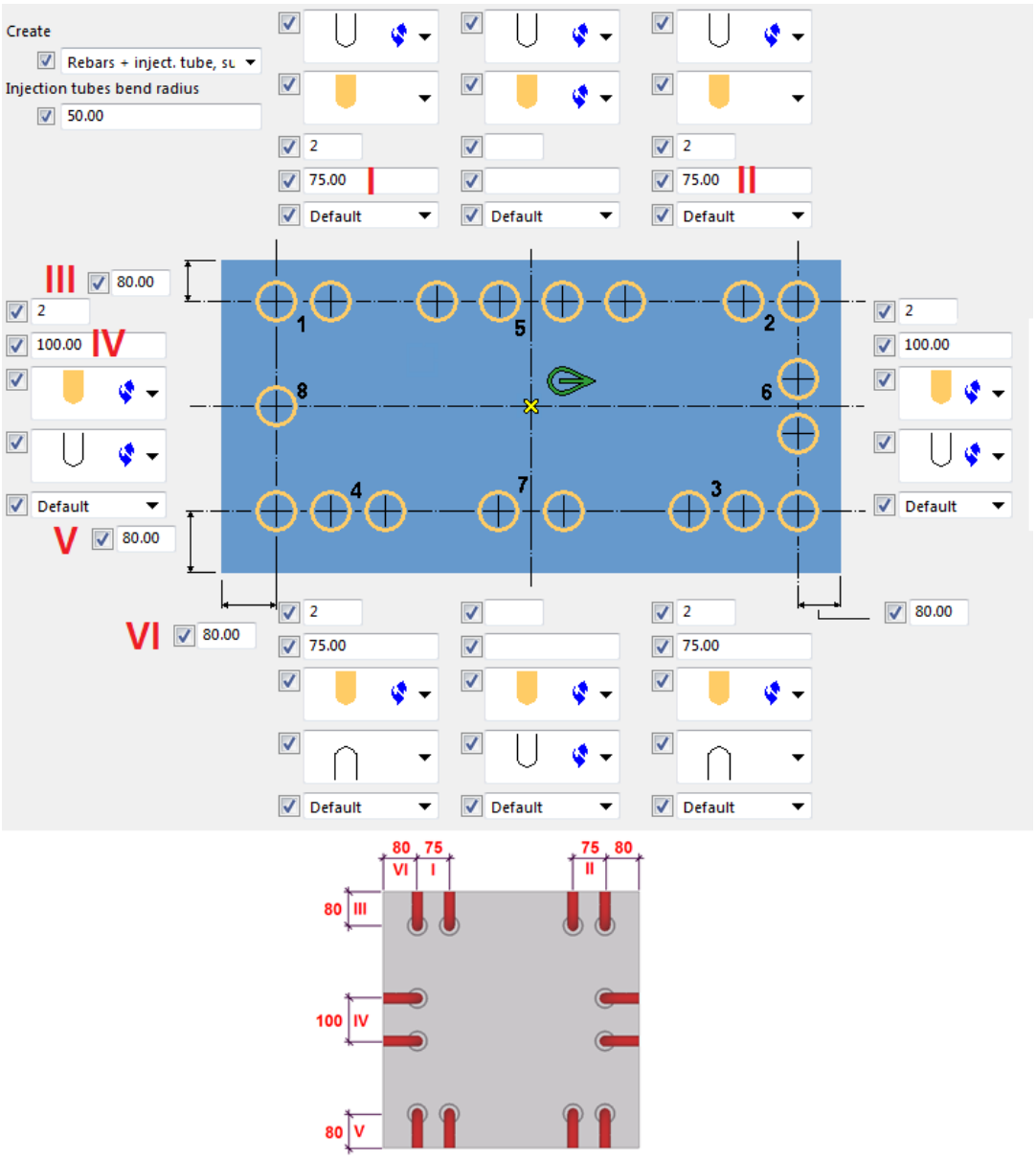
Injection tube properties



	Description
1	Select the direction of the bent bars.
2	Select the direction of the injection hoses.
3	Number of injection tubes.
4	Center-to-center distance between the injection tubes.
5	Define the level of the curved injection parts. This is useful if the curved parts are pointing to the same direction. You can define the levels on the **Picture** tab.

	Description
	 Example:  
6	Define the distance from the center of the injection tube to the outer contour of the concrete foundation.

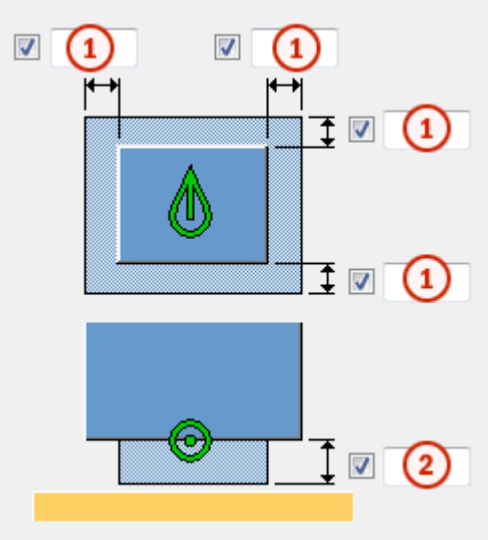
Example



Column tab

Use the **Column** tab to control the dimensions of the column cut-out.

Column dimensions

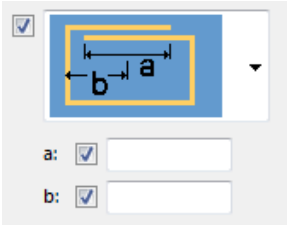


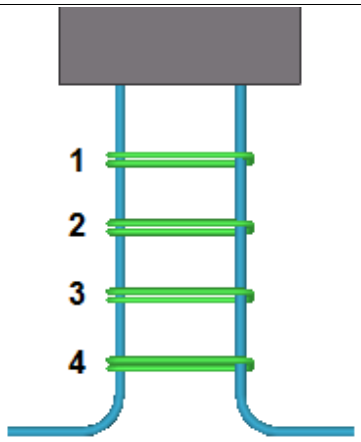
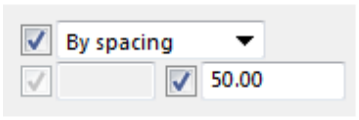
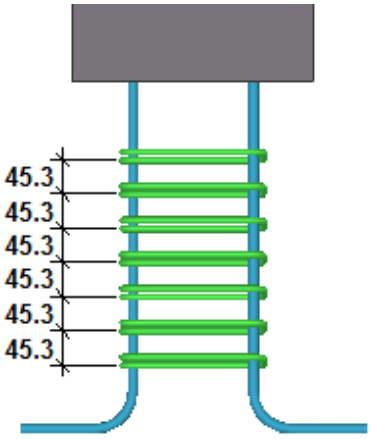
	Description
1	Size of the column cut-out.
2	Height of the column cut-out.

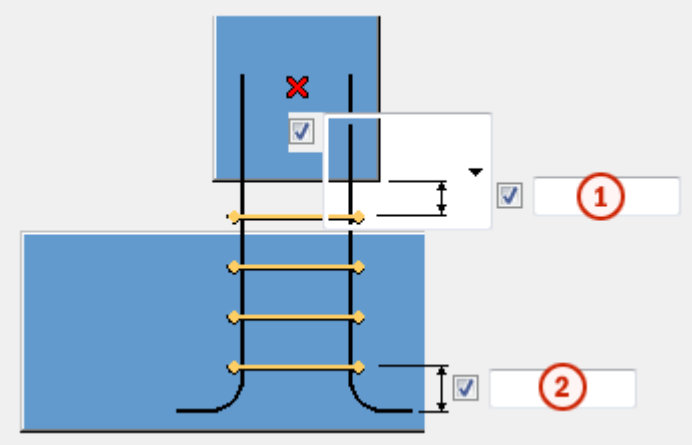
Stirrups tab

Use the **Stirrups** tab to control the stirrup properties.

Stirrup properties

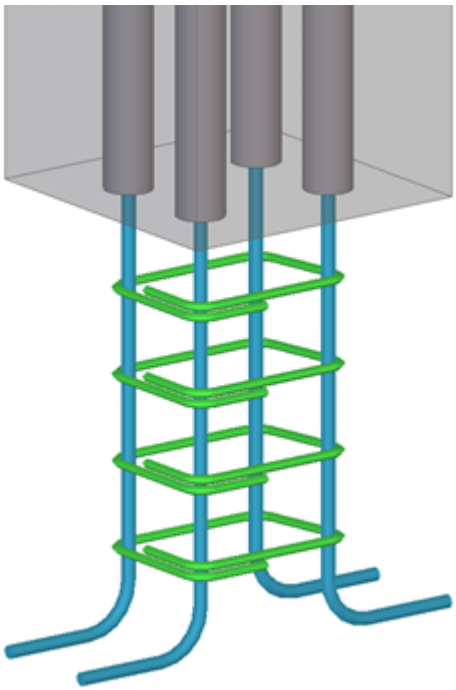
Option	Description
	Select whether the stirrups are created. Define the length of the laps.
Grade	Grade of the stirrups.
Size	Size of the stirrups.
End conditions left	Hook for the start point of the stirrups.
End conditions right	Hook for the end point of the stirrups.
Bend lengths left	Hook length for the start point of the stirrup.
Bend lengths right	Hook length for the end point of the stirrup.
Creation method	Creation method of the stirrups. **Number of bars**: Enter the number of the stirrups.

Option	Description
	   **By spacing**: Enter a spacing value.  
	Define the comment, name, class, serie and start number for the stirrups.



	Description
1	Select the start point of stirrups, either from the bottom of the column to the first stirrup or from the top of the reinforcing bar to first stirrup.
2	Define the distance from the bottom of the reinforcing bar to the last stirrup.

Example



General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

6.2 Reinforcement

This section introduces components that can be used in reinforcement.

See also

Reinforcement for foundations (page 3837)

Beam, column, and slab reinforcement (page 3860)

Lifting (page 4082)

Reinforcement for foundations

Tekla Structures includes the following components that you can use to automatically create reinforcement for foundations:

- Strip footing reinforcement (75) (page 3837)
- Pile cap reinforcement (76) (page 3841)
- Pad footing reinforcement (77) (page 3847)
- Starter bars for pillar (86) (page 3854)
- Starter bars for footing (87) (page 3856)

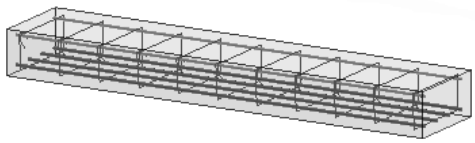
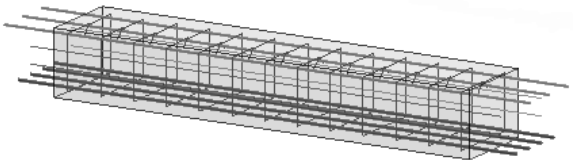
Strip footing reinforcement (75)

Strip footing reinforcement (75) creates reinforcement for a concrete strip footing.

Bars created

- Longitudinal bars for the top and bottom surfaces and sides of the footing
- Stirrups

Use for

Situation	More information
Straight strip footings that have rectangular cross sections	
	Main bars entirely inside the footing, no side bars, stirrup laps at stirrup corners.
	Main bars protruding from the footing, two bars on both sides, stirrup laps in the middle of the top surface.

Do not use for

Footings that have:

- Irregular cross sections
- Skew or cut corners

Before you start

- Create the concrete strip footing.
- Calculate the required area of reinforcement.

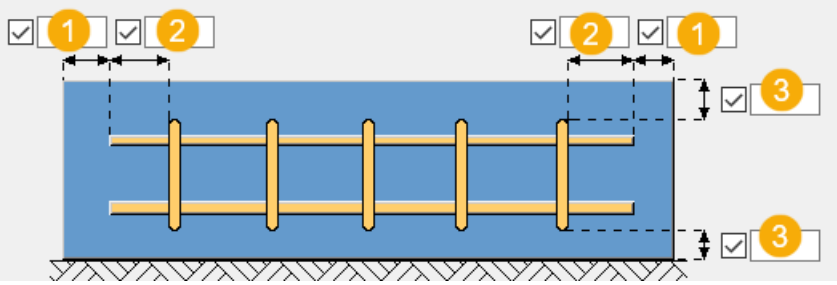
Selection order

1. Select the concrete strip footing.

Picture tab

Use the **Picture** tab to define the concrete cover thickness and stirrup offset.

Cover thickness



	Description
1	Cover thickness (strip ends)
2	Stirrup offset
3	Cover thickness (top and bottom)

Main bars tab

Use the **Main bars** tab to define the properties of top, bottom, left and right bars.

Bond length of main bars

Bond lengths define how far main bars extend into adjacent structures at the ends of strip footings. Use the **Bond length 1** boxes for the first end of the footing (with the yellow handle), and the **Bond length 2** boxes for the second end of the footing (with the magenta handle).

You can define bond lengths separately for:

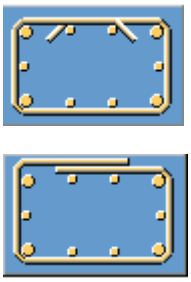
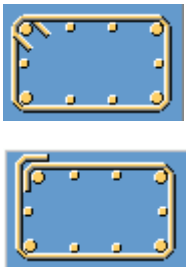
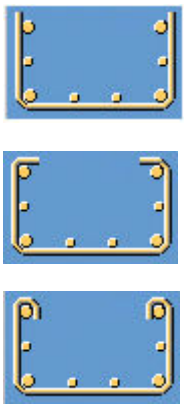
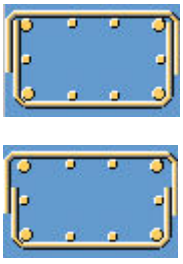
- Top bars
- Bottom bars

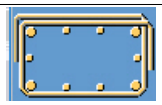
- Bars on the left side of the footing
- Bars on the right side of the footing

Stirrups tab
Use the **Stirrups** tab to define stirrup properties and spacing type.

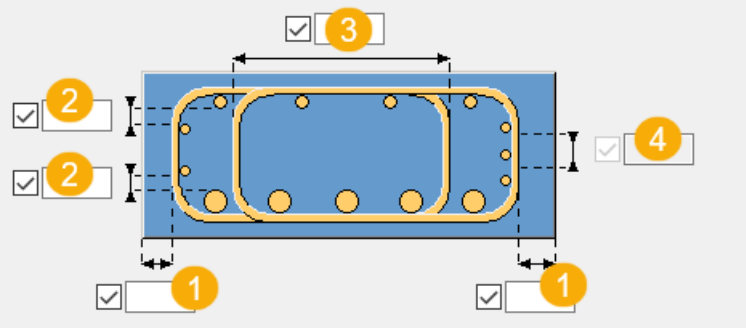
Bend type

Select the location of the stirrup laps in the strip footing.

	Option
At mid	
At corner	
U-shaped	
U-shaped Enter the overlap length in the **Transverse** picture for the first two options.	

	Option
	
Double stirrup bars	
Straight transverse bar	

Stirrup dimensions



	Description
1	Cover thickness (sides)
2	Outer distance between the main bars and outer side bars
3	Overlap length of the double stirrup
4	Overlap length of the double U-bars

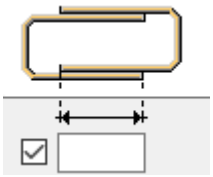
Bending direction

	Option
1	
2	
3	

	Option
4	

End shape of double stirrup bars

If you selected double stirrup bars, you can select the end shapes for bars from the list.

Option	Examples
135-degree Default	
90-degree	
Overlapped If you select overlapped, you can enter the overlap length.	

Attributes tab
Use the **Attributes** tab to define the numbering properties of bars and stirrups.

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Pile cap reinforcement (76)

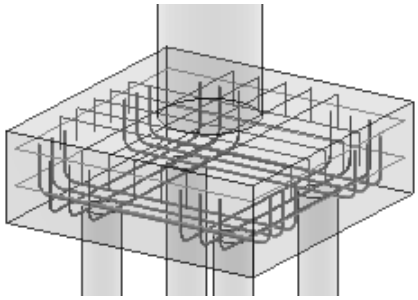
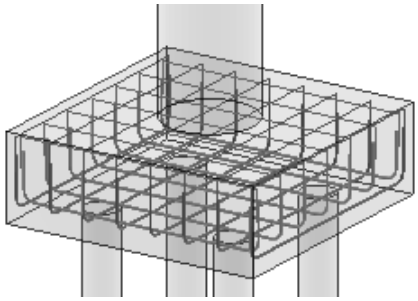
Pile cap reinforcement (76) creates reinforcement for a concrete pile cap.

Bars created

- Bars in two directions for the top and bottom surfaces of the pile cap

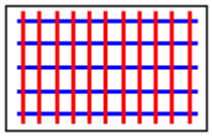
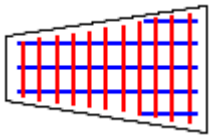
- Lacer bars

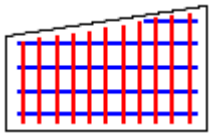
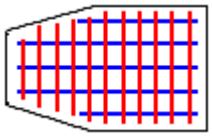
Use for

Situation	More information
	Bottom bars concentrated over the piles, top bars under the column. Two lacer bars.
	Bars evenly distributed on the bottom and top surfaces. No lacer bars.
Rectangular footings with or without cut corners, footings that are skewed on one or both sides	Pad footing and pile cap shapes
Bars on the top, bottom, or both surfaces of the footing	
Straight or bent bar ends	

Pad footing and pile cap shapes

Use the **Pile cap reinforcement (76)** to create reinforcement for the following shapes of foundations:

Shape	Description
	Rectangular
	Skewed on two sides

Shape	Description
	Skewed on one side
	Rectangular with cut corners

Before you start

- Create the concrete pile cap.
- Calculate the required area of reinforcement.

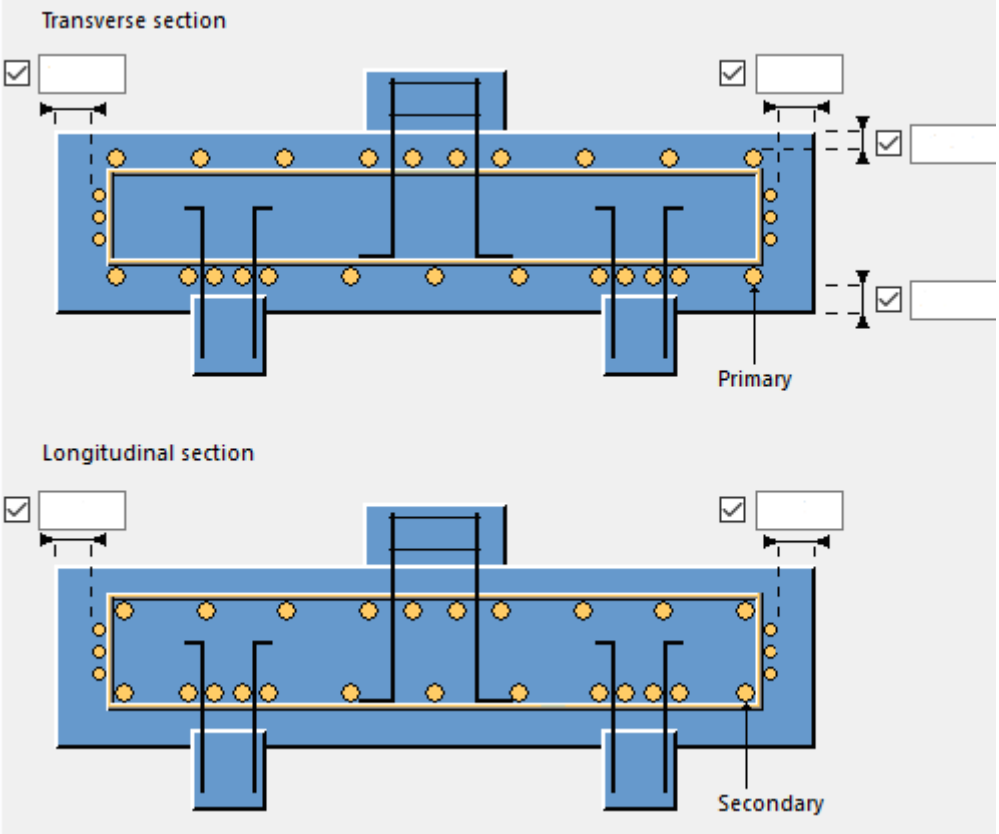
Selection order

1. Select the concrete pile cap.
2. Select the piles and/or columns.
3. Click the middle mouse button to finish.

Picture tab

Use the **Picture** tab to define the concrete cover thickness and primary bar direction.

Enter the concrete cover thicknesses:



Select the primary bar direction to either **Parallel with longer dimension** or **Parallel with shorter dimension**.

Primary/Secondary top bars and Primary/Secondary bottom bars tabs
Use the **Primary/Secondary top bars** and **Primary/Secondary bottom bars** tabs to define the bar properties.

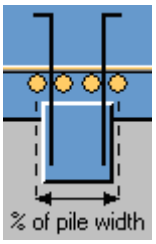
Bar properties

Option	Description
Grade	Strength of the steel used in the reinforcing bar. This field cooperates with the **Size** field.
Size	Diameter of the reinforcing bar. Pressing the **...** button right of the

Option	Description
	field will open the **Select rebar** dialog box. In the dialog box you can select the grade and the accompanying diameter. **NOTE** Selecting a size will override the value in the **Grade** field.
End conditions left/right	Defines the shape of the rebar end. Default value is **Straight**.
Bend lengths left/right	Defines the length of the left/right end-extension.
Creation method	**Number of bars** A fixed number of bars is created. The spacing between the bars is automatically calculated.
	By spacing A accompanying field becomes active. The entered value is the fixed spacing for the rebars. The number of rebars is calculated automatically.

Pile caps

Define the pile cap reinforcement using the following properties:

Field	Description
To suit pile/column	Select **Yes** to concentrate main bars over piles and under columns.
% of pile/column width 	The area where the bars are concentrated, as a percentage of the width of the pile or column. For example, if the pile diameter or width is 500 mm, enter 120 in the **% of pile width** field to concentrate bars in a 600 mm-wide area over the pile.
Bar portioning (%)	The proportion of bars concentrated over a pile or under a column.

Field	Description
Bar portioning (number of bars)	The number of bars concentrated over a pile or under a column.

Lacer bars tab

Use the **Lacer bars** tab to create and define lacer bars.

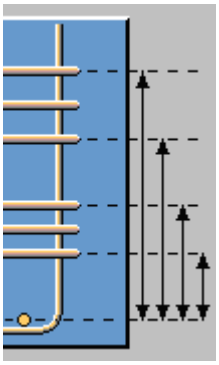
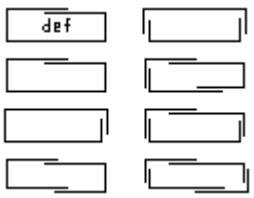
Lacer bars are reinforcing bars that loop around the sides of a concrete footing.

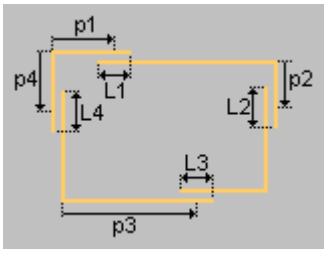
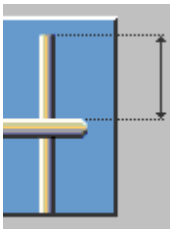
You can create up to six different groups of lacer bars in a footing. Each group can contain different values for:

- Grade
- Bar size
- Number of bars
- Spacing
- Shape
- Dimensions

To create lacer bars for a footing:

1. In the **Lacer bar option** list box, select **Yes** to create lacer bars.
2. Enter properties for each lacer bar group:

Property	Description
	Quantity, spacing, and location of lacer bar groups. Tekla Structures only uses information from some of the fields, in this order of priority: 1. **Number of bars** and **Spacing** 2. **Number of bars**, **Start**, and **End** 3. **Spacing**, **Start**, and **End**
Type	The number and location of laps. The options are: 

Property	Description
Orientation	The options are **Default**, **Front**, and **Back**.
p1...p4 **L1...L4**	The exact locations and lengths of lacer bar laps. Locations are measured from the corner of the bar to the midpoints of laps. 
	The location of the topmost lacer bar, measured from the end of main bars. Entering a value here overrides the location defined in the **End** field.

Attributes tab
Use the **Attributes** tab to define numbering properties of bars.

Option	Description
Prefix	Prefix for the bar position number.
Start number	Start number for the bar position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

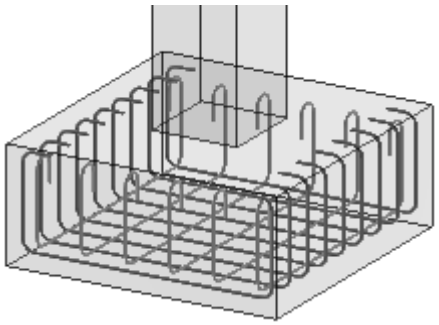
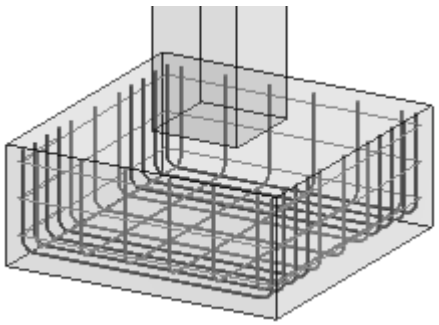
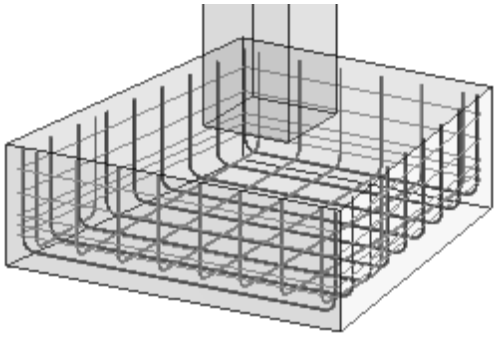
Pad footing reinforcement (77)

Pad footing reinforcement (77) creates reinforcement for a concrete pad footing.

Bars created

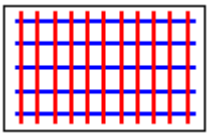
- Bars in two directions for the bottom surface of the pad footing
- Lacer bars

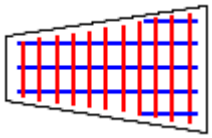
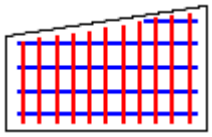
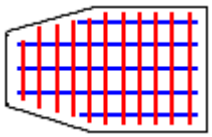
Use for

Situation	More information
	Rectangular footing, 90-degree hooks at the primary bar ends, 180-degree hooks at the secondary bar ends, no lacer bars.
	Rectangular footing, three zones of primary bars with different spacing, straight bar ends, three lacer bars.
	Footing skewed on two sides, two groups of lacer bars with different spacing.
Rectangular footings with or without cut corners, footings that are skewed on one or both sides	Pad footing and pile cap shapes

Pad footing and pile cap shapes

Use the **Pad footing (77)** to create reinforcement for the following shapes of foundations:

Shape	Description
	Rectangular

Shape	Description
	Skewed on two sides
	Skewed on one side
	Rectangular with cut corners

Before you start

- Create the concrete pad footing.
- Calculate the required area of reinforcement.

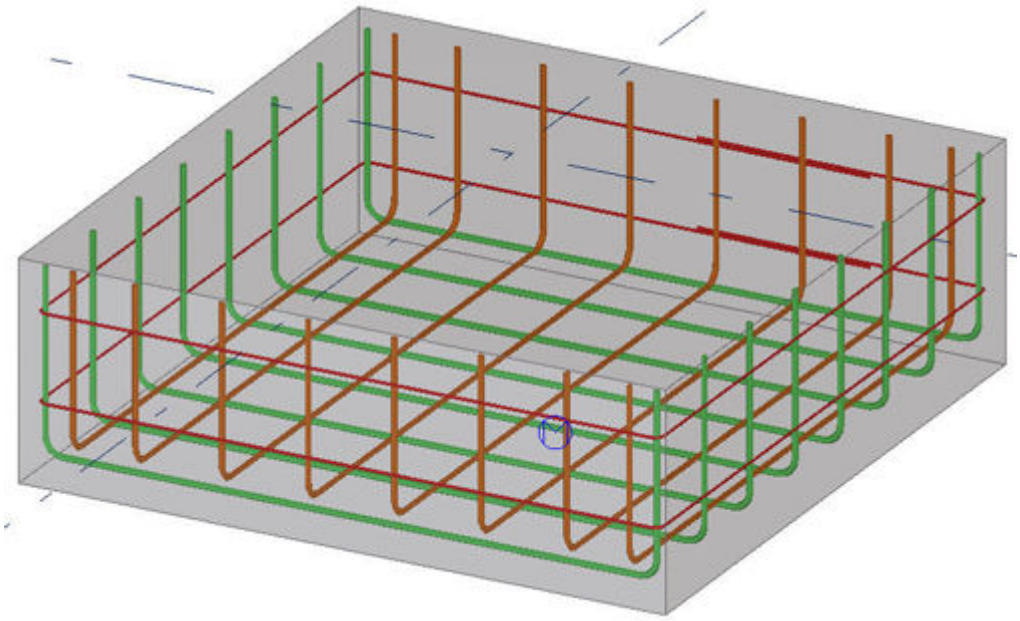
Selection order

1. Select the concrete pad footing.

Add a pad footing reinforcement using Pad footing reinforcement (77)

1. Create a pad footing.
2. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
3. Enter `pad footing` in the search box.
4. Select **Pad footing reinforcement (77)**.
5. Select the pad footing.

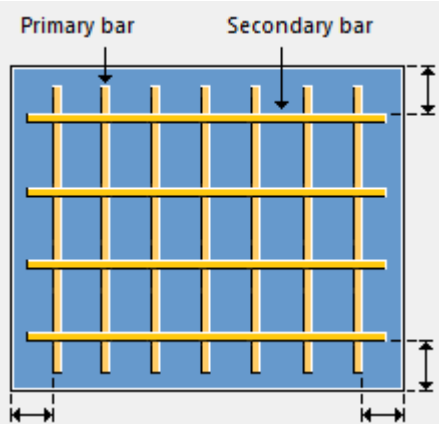
Tekla Structures inserts the lacer bar and bottom reinforcement in the pad footing.

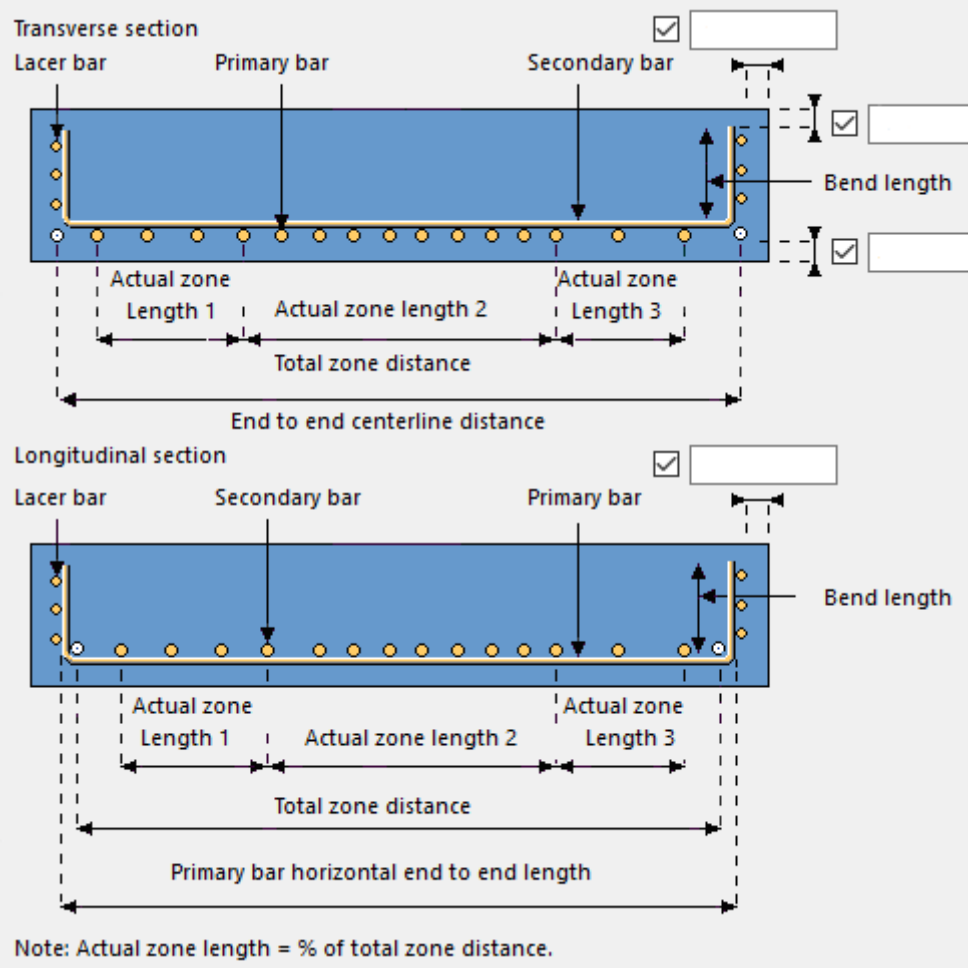


Picture tab

Use the **Picture** tab to define the conrete cover thickness.

Enter the concrete cover thicknesses on the plane and from the plane.





Mesh alignment

Option	Description
Square mesh	Set the alignment of the secondary bars of the created mesh perpendicular to the primary bars.
Mesh aligned to both sides	Align the primary and secondary bars to the skewed edges.

Primary/Secondary bar tab
Use **Primary/Secondary bar** tabs to define the bar properties.

Pad footings

In pad footings, you can arrange the main reinforcing bars into:

- One zone of bars that have the same bar properties

- Three zones of bars that have different bar properties

You set the options separately for primary and secondary bars. Select an option from the **Arrangement** list box.

Hooks

Select to create hooks from **Left end hook** and **Right end hook**.

You can create the hooks as custom hooks from dropdown list.

Lacer bar tab

Use the **Lacer bar** tab to define the lacer bar properties.

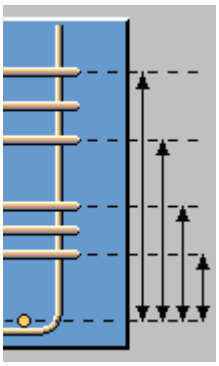
Lacer bars are reinforcing bars that loop around the sides of a concrete footing.

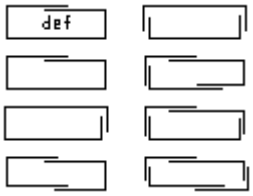
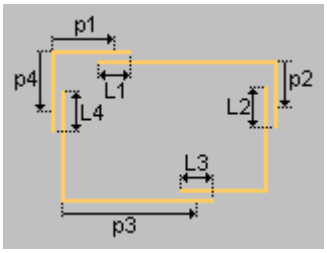
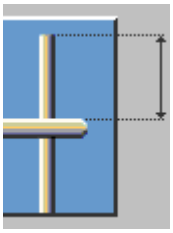
You can create up to six different groups of lacer bars in a footing. Each group can contain different values for:

- Grade
- Bar size
- Number of bars
- Spacing
- Shape
- Dimensions

To create lacer bars for a footing:

1. Open the footing reinforcement properties dialog box and click the **Lacer bars** tab.
2. In the **Lacer bar option** list box, select **Yes** to create lacer bars.
3. Enter properties for each lacer bar group:

Property	Description
	Quantity, spacing, and location of lacer bar groups. Tekla Structures only uses information from some of the fields, in this order of priority: 1. **Number of bars** and **Spacing** 2. **Number of bars**, **Start**, and **End** 3. **Spacing**, **Start**, and **End**

Property	Description
Type	The number and location of laps. The options are: 
Orientation	The options are **Default**, **Front**, and **Back**.
p1...p4 **L1...L4**	The exact locations and lengths of lacer bar laps. Locations are measured from the corner of the bar to the midpoints of laps. 
	The location of the topmost lacer bar, measured from the end of main bars. Entering a value here overrides the location defined in the **End** field.

Attributes tab
Use the **Attributes** tab to define numbering properties.

Field	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

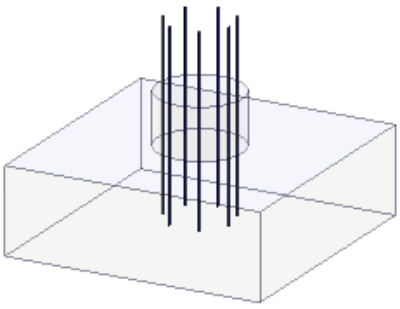
Starter bars for pillar (86)

Starter bars for pillar (86) creates starter bars in a footing and a pedestal for a column. Starter bars may go through a pedestal or straight to the column. Starter bars can be in a rectangular or circular form.

Bars created

- Starter bars (straight or L-shaped)
- Stirrups (optional)

Use for

Situation	More information
	**Starter bars for pillar (86)** creates straight or L-shaped starter bars that go through a rectangular or circular pedestal. Starter bars can have stirrups.

Before you start

- Create the footing.
- Calculate the required area of reinforcement.

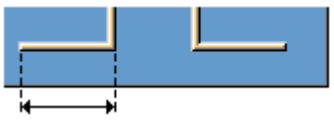
Selection order

1. Footing
2. Pedestal

Picture tab

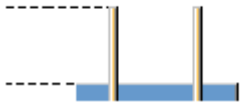
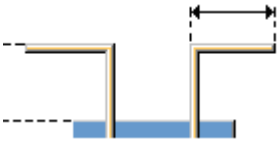
Use the **Picture** tab to define bar dimensions and locations, spacing of bars, stirrup type, and concrete cover thickness.

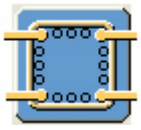
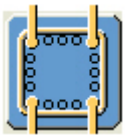
Select the shape of the starter bars:

Option	Description
	L-shaped starter bars. Define the length of the bar.

Option	Description
	Straight starter bars.
	L-shaped starter bars. Define concrete cover thickness.

Select the shape for the top of the starter bars:

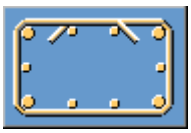
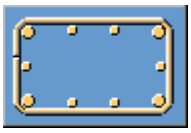
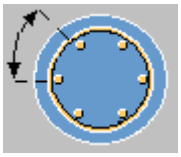
Option	Description
	The top of the starter bars is straight. Default
	The top of the starter bars is bent. Define the length of the bent part.

Option	Description
Number of stirrups/Spacing	Define the number of stirrups and their spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.
Create corner bars	Select to create corner bars. Default value is **Yes**.
 	Select the placement of the corner bars.
Grade	Define the grade of the reinforcing bar.
Size	Define the size of the reinforcing bar.

Option	Description
Bending radius	Define the bending radius of the reinforcing bar.

Stirrups

Use these options to define stirrup laps in footings:

Option	Description
	Laps on the side of the stirrups 45-degree hooks at bar ends
	Laps at stirrup corners 135-degree hooks at bar ends
	Laps at stirrup corners 90-degree hooks at bar ends
	If the starter bars are in a circular form, you must define the angle of stirrup overlap.

Parameters tab

Use the **Parameters** tab to define numbering properties of bars.

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

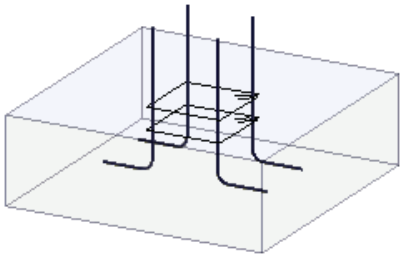
Starter bars for footing (87)

Starter bars for footing (87) creates starter bars in a footing for a column. Starter bars may go through a pedestal or straight to the column. Starter bars can be in a rectangular or circular form.

Bars created

- Starter bars (straight or L-shaped)
- Stirrups (optional)

Use for

Situation	More information
	**Starter bars for footing (87)** places starter bars in a footing in rectangular or circular form. Starter bars can be straight or L-shaped, and can have stirrups.

Before you start

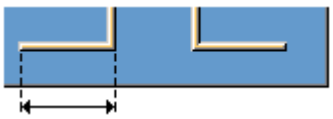
- Create the footing.
- Calculate the required area of reinforcement.

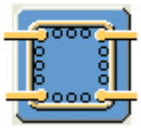
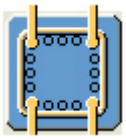
Selection order

1. Footing

Picture tab
Use the **Picture** tab to define bar dimensions and locations, number and spacing of bars, and concrete cover thickness.

Select the shape of the starter bars:

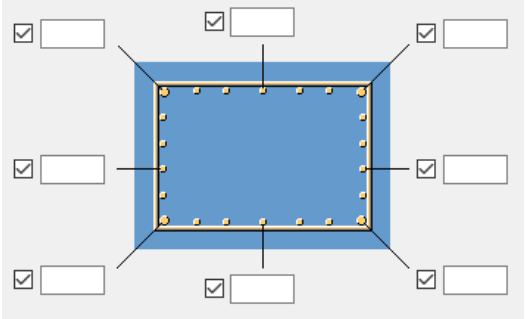
Option	Description
	L-shaped starter bars. Define the length of the bar.
	Straight starter bars.
	L-shaped starter bars. Define concrete cover thickness.

Option	Description
Number of stirrups/Spacing	Define the number of stirrups and their spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.
Create corner bars	Select to create corner bars. Default value is **Yes**.
 	Select the placement of the corner bars.
Grade	Define the grade of the reinforcing bar.
Size	Define the size of the reinforcing bar.
Bending radius	Define the bending radius of the reinforcing bar.

Bottom cover tab

Use the **Bottom cover** tab to define bottom cover thickness for bar groups.

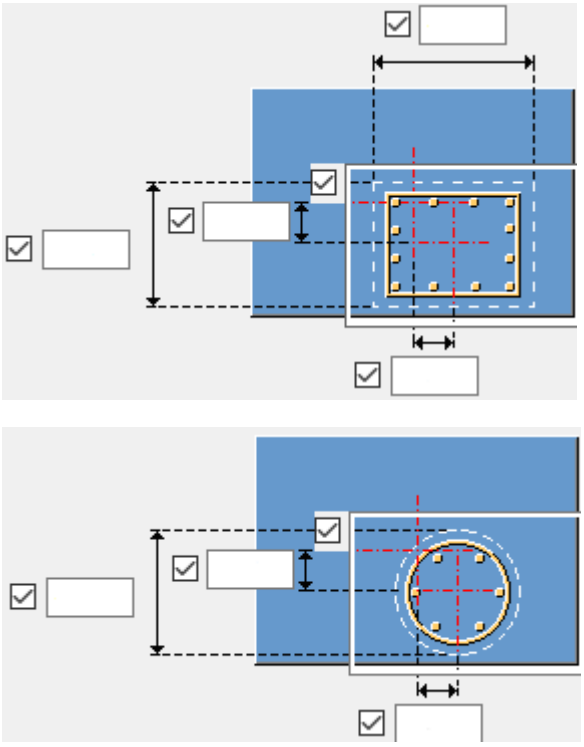
Select to set bottom cover thickness from the **Same for all** list:

Option	Discription
Yes	Sets the same bottom cover thickness for all corner and side bars.
No	Enter the separate bottom cover thickness for each bar group. 

Location tab
Use the **Location** tab to define starter bar location and stirrup type.

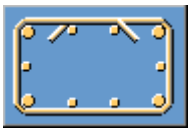
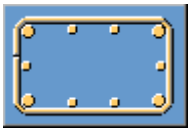
Starter bar location

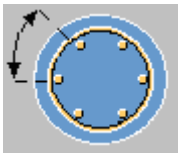
Enter the distance from the center of the bar group to the center of the footing, in two directions:



Stirrups

Use these options to define stirrup laps in footings:

Option	Description
	Laps on the side of the stirrups 45-degree hooks at bar ends Default
	Laps at stirrup corners 135-degree hooks at bar ends
	Laps at stirrup corners 90-degree hooks at bar ends

Option	Description
	If the starter bars are in a circular form, you must define the angle of stirrup overlap.

Parameters tab
Use the **Parameters** tab to define numbering properties of bars.

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Beam, column, and slab reinforcement

Tekla Structures includes the following components that you can use to automatically create reinforcement for beams, columns, and slabs:

- Detailing manager (page 3861)
- Mesh Bars / Mesh Bars by Area (page 3863)
- Rebar coupler and anchor tools (page 3875)
- Slab bars (18) (page 3890)
- Beam reinforcement (63) (page 3892)
- Stirrup reinforcement (67) (page 3898)
- Longitudinal reinforcement (70) (page 3904)
- Beam end reinforcement (79) (page 3907)
- Corbel reinforcement (81) (page 3912)
- Round column reinforcement (82) (page 3919)
- Rectangular column reinforcement (83) (page 3929)
- Hole reinforcement for slabs and walls (84) (page 3943)
- Hole creation and reinforcement (85) (page 3949)

- Braced girder (88) (page 3953)
- Braced girder (89) (page 3974)
- Reinforcement mesh array in area (89) / Reinforcement mesh array (91) (page 3995)
- Rectangular area reinforcement (94) (page 4000)
- Wall panel reinforcement / Double wall edge and opening reinforcement (page 4015)
- Multiple Wire Size Mesh (page 4026)
- Embedded anchors (8) (page 4030)
- Embed (1008) (page 4070)
- Continuous Beam Reinforcement (page 4080)

Detailing manager

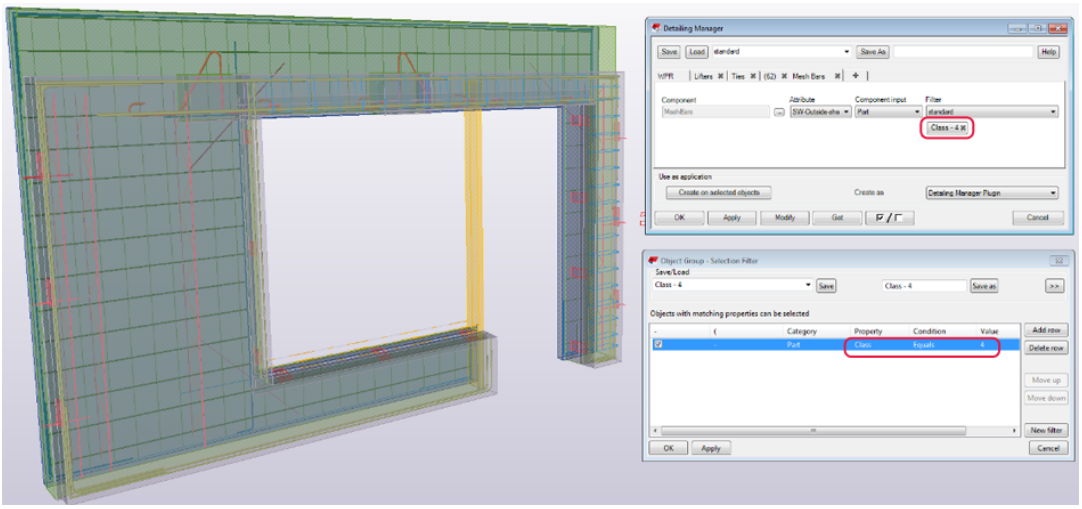
Detailing manager is used to apply detailing components into any structure. With **Detailing manager** you can define rules to apply multiple components at one go to detail the structure of a cast unit or the entire model.

You can use selection filters to define detailing rules for automating repetitive detailing tasks. You can add custom details, extensions or system components with **Detailing manager**.

You have three options:

- Use **Detailing manager** to have a part as input. All parts in the cast unit are examined and detailed by the set rules.
- Use **Detailing manager** to have a cast unit as input. All parts in the cast unit are examined and detailed by the set rules.
- Use **Detailing manager** to have a component as input, for example, tools such as **Floor layout** or **Wall layout** where the same tool creates multiple parts. All parts in the component are examined and detailed by the set rules.

The image below shows an example of **Detailing manager**. In the example, the **Mesh bars** component is created to the outside shell of the cast unit because it has the class 4 filter selected and there is a selection filter for objects with class 4.



Properties

Option	Description
Rule	Use the rules to define detailing settings. The maximum number of rules is 25. Click + to add a new rule. You can rename a rule by double-clicking the rule tab. You can remove the content from a rule tab by double-clicking the rule tab and selecting **Clear**. You can remove rule tabs by double-clicking the rule tab and selecting: • **Close** to remove the selected tab. • **Close other tabs** to remove other tabs than the selected tab. • **Close all tabs** to remove all tabs. **Detailing manager** adds a new empty rule tab. You can also remove rules by clicking ✖ on a rule tab.
Component	Click ▭ to select the component that you want to use to detail a structure. Double-click the component in the **Applications & components** catalog to add it to the **Component** box.
Attribute	Select the attribute file for the component. If you do not select a file, the standard file is used.
Component input	Define the creation method of the selected component: • **Part**: Select the part to which you are creating the component.

Option	Description
	• **Part + one point**: Select the part and one additional point on the part. With this option, **Detailing manager** places the center of the object boundary box to that point. Use this setting with custom component details. • **Part + two points**: Select the part and two additional points on the part. With this option, **Detailing manager** uses the start and the end point of the part as insertion points.
Filter	Select which parts inside the selected cast unit or component get the selected component. The available selection filters are listed in the filter list. You can also create new selection filters. You can use five selection filters if you want to create an intersection of the filters.
Create on selected objects	**Detailing manager** is created to selected parts, cast units or components. The rule settings that are currently set in the **Detailing manager** dialog box are used.
Create as	• **Detailing manager plug-in**: Components are inserted as **Detailing manager** plug-ins and not as separate components. Select this option if you use a component as input. • **Separate components**: With this setting, the inserted components do not have any connection to **Detailing manager**.

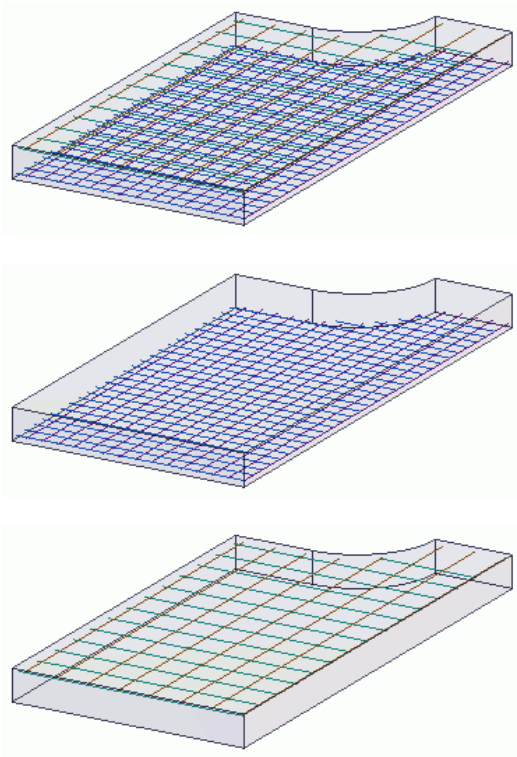
Mesh Bars / Mesh Bars by Area

Mesh Bars and **Mesh bars by area** create reinforcement for concrete slabs or walls.

Objects created

- Primary bars
- Crossing bars

Use for

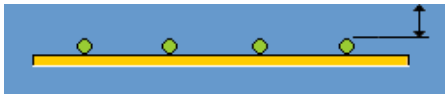
Situation	Description
	Reinforcement for the bottom or the top surface of the concrete element, or for both.

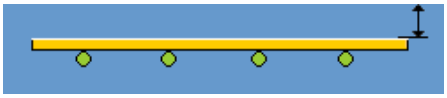
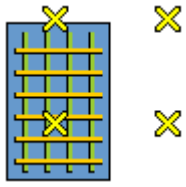
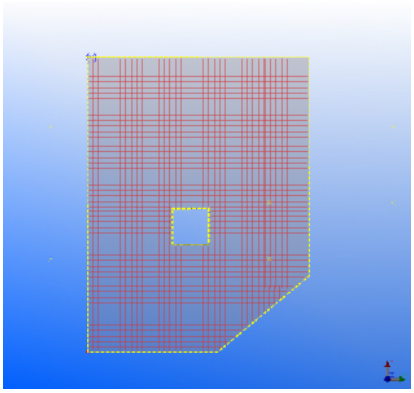
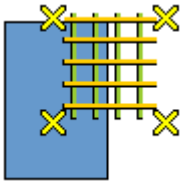
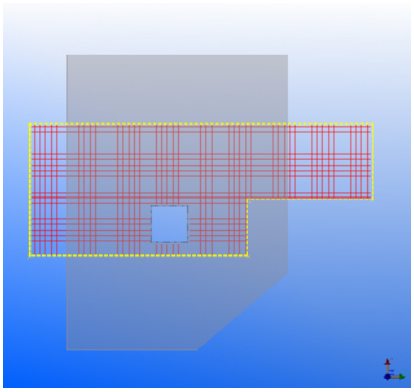
Selection order

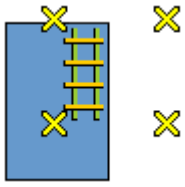
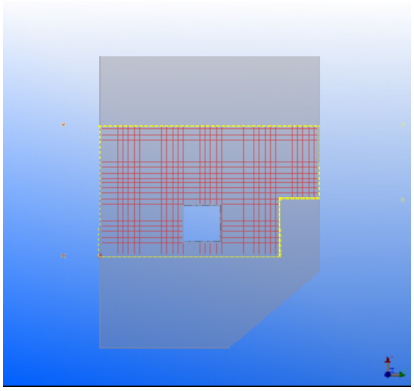
Component	To create mesh bars
Mesh Bars	1. Select a concrete slab. The reinforcement is created automatically.
Mesh bars by area	1. Select a concrete slab. 2. Select a group of points to define a working polygon. The reinforcement is created automatically.

Picture tab

Use the **Picture** tab to define how the top and bottom bars are created.

Option	Description	
Create bars		Top bars with primary bars above secondary bars

<table>
<tr><th>Option</th><th colspan="2">Description</th></tr>
<tr><td></td><td>  </td><td>Top bars with secondary bars above primary bars</td></tr>
<tr><td></td><td> </td><td>No bars</td></tr>
<tr><td></td><td colspan="2">The same options are available for the bottom bars as well.
The value defined in the box is the cover thickness for the top/bottom layer.
If you only want to create bars in the middle of the slab, select the No bars option either for the top or bottom bars, and then select Yes in the Place central list that appears.</td></tr>
<tr><td>Mesh area perimeter</td><td colspan="2">This option is only available for Mesh bars by area.
Select the outline that the mesh follows.
The meshes in the examples are created by picking the same points but with different mesh area perimeters.</td></tr>
<tr><td></td><td> 
Part</td><td>Part, for example:
  </td></tr>
<tr><td></td><td> 
Polygon</td><td>Polygon, for example:
  </td></tr>
</table>

Option	Description	
	 Polygon + part	Part + polygon, for example: 
Same as bottom	Select whether the top bars are created using the same properties as the bottom bars. If you select **No**, enter the top bar properties.	

Bar type	Select whether the bars are created as a bar group or as a mesh.
	Depending on the option you select, some other settings and options are available. For example:
	• If you select the **Bar group** option:
	  • You can move, copy, and delete bars using direct modification.
	  To move a bar, drag it to a new location.
	  To copy a bar, hold down **Ctrl** and drag the bar.
	  To delete a bar, select it and press **Delete**.
	  • You can use the grid, minimum length, **Bar grouping**, and **Bar behavior at cuts** settings on the **Detailing** tab.
	• If you select the **Mesh** option, bars are always cut by the cuts in the concrete part.	

Size	Select the diameter of a bar. You can set the size separately for the top primary and secondary bars, and for the bottom primary and secondary bars.	
Grade	Define the strength of the steel used in the bars. You can set the grade separately for the top primary and secondary bars, and for the bottom primary and secondary bars.	

Option	Description
Spacing type	Select whether the spacing is based on the number of bars or on the spacing values. • **By spacings**: enter the distance between the bars in the **Spacing** box. • **Number of bars**: enter the number of bars in the **Number of bars** box. • **By exact spacings**: list the distances between the bars in the **Spacing** box. For example: `200, 200, 50, 50, 50, 100` or `2*200, 3*50, 100`
Spacing	Distances between the bars. Tekla Structures calculates the number of bars. You can set the spacing separately for the top primary and secondary bars, and for the bottom primary and secondary bars.
Number of bars	Number of bars. Tekla Structures calculates the distance between bars. You can set the number of bars separately for the top primary and secondary bars, and for the bottom primary and secondary bars.
Primary bars direction	The direction of the primary bars. Use to change the bar direction. If the **Auto** option is selected, the primary bars are created along the two longest parallel sides of a slab. If there are no parallel sides, you can select to create the bars along the slab x direction instead. To rotate the bars, select one of the direct modification options. For non-perpendicular meshes, use the flexible angle option to individually rotate both arrow sides around their axis.
Up direction	Select which sides are regarded as the top and bottom sides of the slab. If the **Auto** option is selected, then the bars are created on the side with the greatest area.

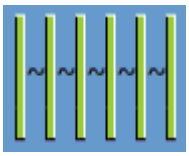
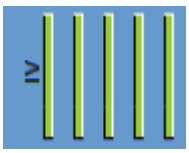
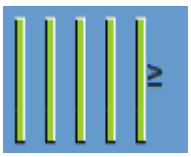
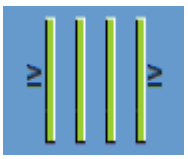
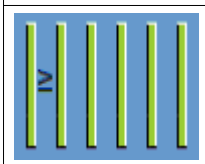
Detailing tab

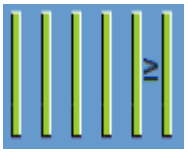
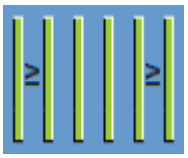
Use the **Detailing** tab to control how the bars are distributed.

Bars that are created as bar groups can be distributed **By grid** or **Without grid**.

Using a grid makes it easier to accurately place bars only at set intervals. When direct modification is switched on and you move or copy bar group bars, they snap to the grid.

Detailing without grid

Option	Description	
Bar distances	Select how the distances between the bars are measured. • **From center of bar** • **From edge of bar** Select whether the distances are the same or different for the top and bottom bars. Using different distances helps in placing top and bottom bars so that their hooks do not collide, for example.	
Adjustment	Select how the bars are distributed. The same options are available for both the primary and the secondary bars.	
		Equal distribution by target spacing value.
		By exact spacing value with flexible first space without first bar.
		By exact spacing value with flexible last space without last bar.
		By exact spacing value with flexible first and last space without first and last bars.
		By exact spacing value with flexible first space.

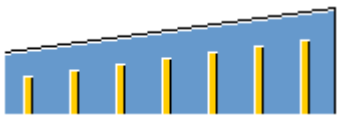
Option	Description	
		By exact spacing value with flexible last space.
		By exact spacing value with flexible first and last space.

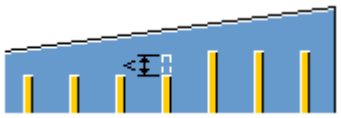
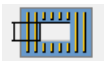
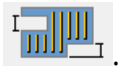
Detailing by grid

The same options are available for both the longitudinal and the crossing bars.

Option	Description
Min overhang	Minimum extension of the longitudinal or crossing bars over the outermost bars of the other direction. You can set the minimum overhang separately for the start and end of the bars.
Min spacing	Minimum distance between the bars.
Longitudinal grid size **Crossing grid size**	Defines the location intervals to which the bar group bars snap when they are moved or copied using direct modification.

Other detailing options

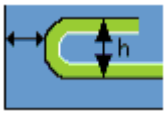
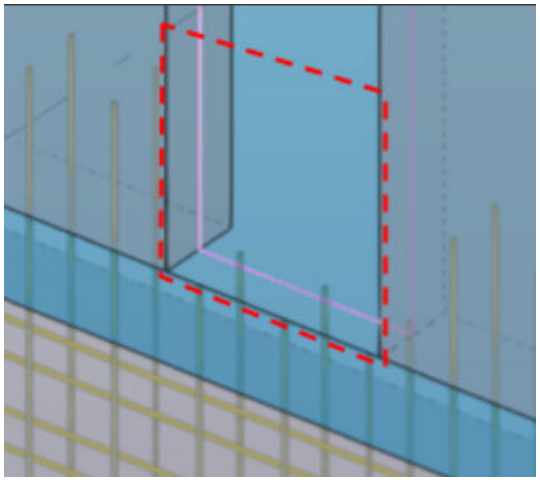
Option	Description	
Minimum length of primary bar to be created **Minimum length of secondary bar to be created**	In bar groups, bars that are shorter than this value are not created.	
Bar grouping	For bar groups, select whether tapered bars are grouped or not. If you group the tapered bars, select how the bars along the tapered edge are handled.	
		Tapered bars are handled normally.

Option	Description	
		Creates as many bars as possible with same length at the tapered edge. The value you enter is the maximum allowed shortening of a bar.
Bar behavior at cuts	Select how bars next to openings are handled. • **Cut bars**: Select whether the bars are cut and which cuts are ignored. In bar groups, you can ignore cuts by part name, part class, or selection filter. With the **Yes, but ignore cuts by filter** option, the **Selection filter** list only shows the selection filters where one rule has the object type 11 (polygon cut) and the other rules have been defined by using the **Template** category. • **Ignore openings smaller than**: The minimum length of a cut (in the direction of the bars) that will cut the bars. Lower values are ignored. This setting only affects bars that are created as bar groups. Bars that are created as a mesh are always cut by the cuts in the concrete part. • **Cover thickness**: Distance between the bar end/start point to the edge of the opening. You can use this option to define the cover thickness for both windows  and doors  . This option is useful if window and door openings need to have different cover thicknesses. • **Create cuts around selected parts**: Select whether the bars are cut by the parts that clash with the reinforced part. The cutting parts are defined by a selection filter.	

Bar end conditions tab

Use the **Bar end conditions** tab to control the cover thickness and bar hooks.

Option	Description
Cover thickness on sides	• **Different each side**: Each end of a bar group can have different distances from part edges. • **Same all sides**: All ends of the bar groups have the same distance from part edge.

Option	Description	
Use bar end conditions around holes	Select whether the same bar end conditions are used around the openings as at part edges.	
End conditions		Creates a straight bar without a hook at the given end of a bar.
		Creates a 90° hook at a given end of bar.
		Creates a 135° hook at a given end of bar.
		Creates a 180° hook at a given end of bar.
		Creates a hook with a free angle at the given end of a bar.
		Creates a double bent hook at the given end of a bar.
Cut bars by negative neighbor parts	Select whether the bars that protrude from the selected concrete parts are cut at the openings of neighbor parts. To cut the bars at the openings, select **Yes**, and then enter the cover thickness. 	
Bending radius	Bending radius at bar ends	

Option	Description
Bending length	Bending length at bar ends

Splicing tab
Use the **Splicing** tab to control the splicing of the reinforcing bars.

Option	Description
Splice bars	Select whether the reinforcing bars are spliced or not.
Manufacturer	Select the manufacturer of the reinforcement. The maximum bar lengths and lap lengths are then listed by the grade and size of the bar. If needed, you can modify these splicing definitions in the `MeshBarsSplicing_Manufacturers.dat` file. You can also copy the default file from `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\system`, edit it, and save it to your project or firm folder.
Maximum length of bars	Maximum reinforcing bar length after which the bars are spliced.
Lapping length	Length of the lapping connection.
Splicing in same cross section	Select how many reinforcing bars can be spliced in the same location. • **1/1** = all reinforcing bars are spliced in the same cross section. • **1/2** = every second reinforcing bar is spliced in the same cross section. • **1/3** = every third reinforcing bar is spliced in the same cross section. • **1/4** = every fourth reinforcing bar is spliced in the same cross section.
Splicing symmetry	Select the symmetry that is applied when the reinforcing bars are spliced. • **Not Symmetrical**: The splice pattern of the reinforcing bar is not symmetrical and the uneven length is only at one side. • **Symmetrical with different lengths at sides**: The splice pattern of the reinforcing bar is symmetrical with uneven lengths at the sides.

Option	Description
	• **Symmetrical with different length at center**: The splice pattern of the reinforcing bar is symmetrical with uneven length in the center.
Splicing offset	Offset of the splice center point from the point where the reinforcing bars originally met.
Minimum splitting distance	Minimum longitudinal distance between two splices in consecutive bars.
Splicing type	Select the type of the splice.
Bar position	Select whether the lapping bars are on top of each other or parallel to each other.

Attributes tab

Use the **Attributes** tab to control the bar group or mesh properties, and user-defined attributes.

Properties

Option	Description
Name	Name of the bar group or mesh.
Prefix	Prefix for the position number of the bar group or mesh.
Start number	Start number for the position number of the bar group or mesh.
Class	Class of the bar group or mesh. If you create a mesh, only the class of primary bars is taken into account.

User-defined attributes (UDAs) for reinforcing bars

The UDAs are predefined in the `MeshBars.Udas.dat` file.

Option	Description
Same for all	Select **Yes** to use the UDA for all reinforcing bars. Select **No** to define the UDA separately for all reinforcing bars.
Top bars **Bottom bars**	Enter a UDA value for the top and bottom reinforcing bars.

Rebar assembly tab

Use the **Rebar assembly** tab to add the created reinforcement as a rebar assembly to the cast units.

Create as rebar assembly

You can add the created reinforcement as a rebar assembly to the cast units with a predefined assembly type, name, prefix, and start number.

Option	Description
Create as rebar assembly	Select **Yes** to create all reinforcement as one rebar assembly, and to include it to the cast unit of the input part.
Rebar assembly type	Select the rebar assembly type. If you do not select the type, the default value of the rebar assembly is used.
Name, **Profile**, **Start number**	Define the name, profile, and start number. If you do not define these, the default values of the rebar assembly are used.

Customize the Attributes tab

You can customize the content of the user-defined attributes section on the **Attributes** tab by using the `MeshBars.Udas.dat` file. The file can be located in any of the folders set in the `XS_FIRM`, `XS_PROJECT` or `XS_SYSTEM` advanced options, and in the model folder.

You can use the file to control which user-defined properties of the created reinforcement can be used in **Mesh Bars** and **Mesh Bars by Area**. We recommend that you only use UDAs that are typically common for all reinforcement.

The sample file shown below contains a full description of all the settings and the format of the file. The lines starting with '//' are comment lines.

```
//
// Customized user defined attributes (UDA) for layer parts created by
Mesh Bars component
//
// Each row shall contain 2 or 3 fields separated by tab(s) or semicolon.
// Please note that all uda names shall be unique
//
// Field 1: The data type of the attribute. Valid values are 'distance',
'float', 'option', 'integer' and 'string'
//          compatible with the actual user defined attributes as
specified in objects.inp
// Field 2: UDA name. This is the name of the user defined attribute.
// Field 3: Label. This text is shown in the attributes dialog. It can be
a translatable label (albl_) or any text.
//
string        comment            j_comment
string        USER_FIELD_1        j_user_field_1
string        USER_FIELD_2        j_user_field_2
string        USER_FIELD_3        j_user_field_3
string        USER_FIELD_4        j_user_field_4
```

Rebar coupler and anchor tools

Rebar coupler and anchor tools is a set of four components that help to model and manage

- reinforcing bar couplers that connect two bars
- end anchors that are devices at the unreserved end of the bar.

The following components are part of **Rebar coupler and anchor tools**:

- Rebar coupler
- Rebar end anchor
- Split rebar and add coupler
- Rebar end trimming

In addition, you can manage the user-defined attributes (UDAs) of the **Rebar coupler and anchor tools** with the **Update rebar attributes** component.

All couplers and end anchors are automatically added to the parent part of the reinforcing bar. This allows them to be shown in reports and drawings because they are added to the sub-assembly of the cast unit.

In some cases the reinforcing bars can move outside of the concrete. This may happen with reinforcing bars that are modeled with two points (straight bars and straight bars with hooks). If the reinforcing bars move outside of the concrete, change the cover thickness to either a negative or a positive value depending on what is required.

Two report templates `Rebar Extra Fabrication Length.rpt` and `Rebar Thread Length.rpt` can be used to inquire **Threaded Length** and **Extra Fab. Length** of the reinforcing bars in the BOMs or BOQs.

Rebar coupler

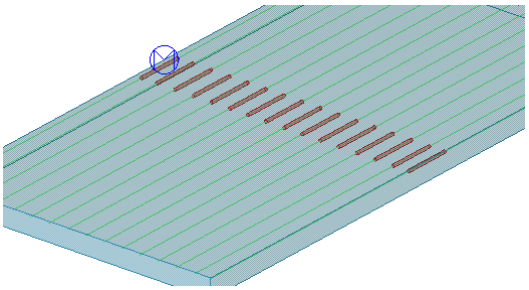
Rebar coupler creates couplers to connect reinforcing bars, or reinforcement meshes, whose bar ends are in contact and parallel.

Objects created

- Round parts

  The parts are created between two reinforcement objects.

Use for

Situation	Description
	Couplers with split reinforcing bars.

Before you start

Couplers can be created between reinforcing bars or reinforcement meshes. The selected reinforcement objects can be of different type and they can even have different number of bars. The only requirement is that one or more bar ends are in contact and parallel. With rebar sets, the couplers can only be created between spliced bars in that particular rebar set.

Selection order

Reinforcing bars or reinforcement meshes	1. Select the primary reinforcing bar(s). 2. Select the secondary reinforcing bar(s).
Rebar sets	1. Select a rebar set. 2. Define a location for a splitter by picking two points. This splitter will become an input object for the couplers. 3. Pick a point. This point defines the side of the primary reinforcing bar(s). This is needed if you have applied different properties for the primary and secondary bar(s). or 1. Select an existing splitter in a rebar set. 2. Pick a point. This point defines the side of the primary reinforcing bar(s). This is needed if you have applied different properties for the primary and secondary bar(s).

Couplers are created at each location where

- the bar ends are parallel enough (angle < 5 degree) and

- the gap along the bar end is less than the length of the coupler and
- the offset perpendicular to the bar end legs is less than the diameter of the bars

When there is no valid and no parallel end-to-end location between the selected bars, the component creates a dummy part that identifies the unsuccessful coupler insertion.

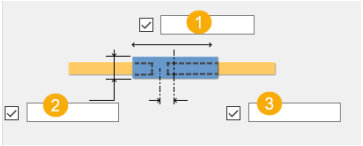
Parameters tab

Use the **Parameters** tab to define the coupler properties.

Option	Description
Use manually entered values	Enter values for the coupler properties.
Use auto attribute file	The values in the auto attribute file override the coupler component values. Typically the auto attributes file contains one or more attribute values to be used with certain bar sizes. Go to Tekla Warehouse to get auto attribute files for your project. For more information on the structure of the auto attribute files, see **Customize attribute files, part mapping and user-defined attributes (UDAs)**.

Option	Description
Use custom component	Select **Yes** to create the coupler as a custom part. Select **No** to create the coupler as a normal part.
Name	Type the name of the custom component, or select it from the **Applications & components** catalog. Ensure that the selected component is a custom part.
Attribute	Type the name of the attribute file of the custom component. The coupler custom part is created using the saved attributes given here.
Input points	The order of the start and the end points of the custom component.

Option	Description
Add to rebar assembly	To add the coupler as a sub-assembly to the rebar assembly, select either **Yes, to primary bar** or **Yes, to secondary bar**. Note that you must first create the rebar assembly, otherwise adding is not possible. The default value is **No**.

Option	Description
Numbering series	Prefix and a start number for the coupler part position number. In the **Inherit from** list, select which numbering settings are used for couplers. • With **None**, enter the part and assembly prefix and start numbers. • With **Rebar**, the numbering settings of the primary bars are used. • With **Cast unit**, the cast unit numbering settings are used.
Attributes	Name, diameter, material, finish and class of the coupler parts.
	**1** The overall end-to-end length of the coupler If you use a custom part, this is the length between the start point and the end point of the custom part. **2** Coupler profile width **3** Coupler offset in relation to the free gap center line. Positive values move the coupler to the right and negative to the left. You can also control this offset in the auto attribute file by using the attribute `CouplerOffset`.

When you create the couplers as custom parts, the properties in the **Numbering series** and **Attributes** sections can be filled from the custom part settings if you have named the properties in a certain way.

Rebar attributes tab

Use the **Rebar attributes** tab to define user-defined attributes (UDAs) for rebars.

Option	Description
Threaded length	The value is written to the reinforcing bar UDA. The fields are used for checking which bars have threads and what are the thread values so that they can be shown in drawings and reports.
Extra fabrication length	This value is written to the reinforcing bar UDA. This value does not affect the length of the reinforcing bar. You need to add this value in your drawings and reports to get the correct length if required.
Method **Type** **Product** **Code**	UDAs written to the reinforcing bars. You can set the reinforcing bar end method and the coupler type, and add a product name and a code for reporting purposes. The used attribute name depends on which end of the reinforcing bar the coupler was created.

NOTE The above user-defined attributes affect numbering. Reinforcing bars with different attribute values get different part marks.

Coupler attributes tab

Use the **Coupler attributes** tab to define user-defined attributes (UDAs) for couplers or end anchor parts. For each UDA, you can select to inherit the value from the rebar or the cast unit. To enter a value, select the inherit option **None**.

Note that if the cast unit does not have any value for that particular UDA, the value is taken from the main part of the cast unit. By default, you can enter a **Comment** and values for **User field 1 ... 4** UDAs. You can customize the set of UDAs by modifying the UDA fields in the `RebarCouplers.Udas.dat` file, located in the `..\Environments\common\system` folder.

Rebar end anchor

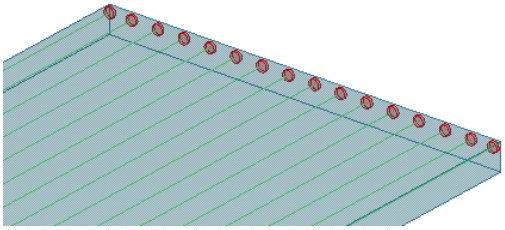
Rebar end anchor creates end anchors at the ends of reinforcing bars or reinforcement meshes.

Objects created

- Round parts

  The parts are created at reinforcing bar ends.

Use for

Situation	Description
	Reinforcing bars with end anchors.

Selection order

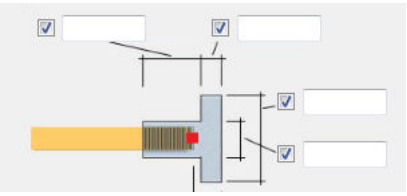
Reinforcing bars or reinforcement meshes	1. Select the reinforcing bar(s). 2. Pick any point close to the bar end where you want to create the end anchors.
Rebar set bars	1. Select a rebar set. 2. Define a location where to add an end detail modifier by picking two points. This end detail modifier will become an input object for the end anchors. or 1. Select an existing end detail modifier in a rebar set.

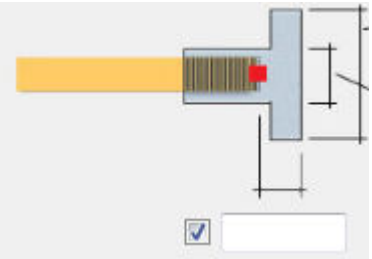
Parameters tab

Use the **Parameters** tab to define the end anchor properties.

Option	Description
Use manually entered values	Enter values for the end anchor properties.
Use auto attribute file	The values in the auto attribute file override the end anchor component values. Typically the auto attributes file contains one or more attribute values to be used with certain bar sizes. Go to Tekla Warehouse to get auto attribute files for your project. For more information on the structure of the auto attribute files, see **Customize attribute files, part mapping and user-defined attributes (UDAs)**.

Option	Description
Use custom component	Select **Yes** to create the end anchor as a custom part. Select **No** to create the end anchor as a normal part.
Name	Type the name of the custom component, or select it from the **Applications & components** catalog. Ensure that the selected component is a custom part.
Attribute	Type the name of the attribute file of the custom component. The end anchor custom part is created using the saved attributes given here.
Input points	The order of the start and the end points of the custom component.
Add to rebar assembly	To add the coupler as a sub-assembly to the rebar assembly, select either **Yes, to primary bar** or **Yes, to secondary bar**. Note that you must first create the rebar assembly, otherwise adding is not possible. The default value is **No**.

Option	Description
Numbering series	Prefix and a start number for the end anchor part position number. In the **Inherit from** list, select which numbering settings are used for couplers. • With **None**, enter the part and assembly prefix and start numbers. • With **Rebar**, the numbering settings of the primary bars are used. • With **Cast unit**, the cast unit numbering settings are used.
Attributes	Name, diameter, material, finish and class of the end anchor parts.
	The dimensions of the end anchor. If you use a custom part, define the length between the start point and the end point of the custom part using these values.

Option	Description
	The offset of the far most point of the coupler measured from the physical end of the reinforcing bar. Note that you cannot control the end of the reinforcing bar with this component. You need to make sure the end concrete cover of the reinforcing bar is greater or equal to this offset plus the minimum concrete cover.

When you create the end anchors as custom parts, the properties in the **Numbering series** and **Attributes** sections can be filled from the custom part settings if you have named the properties in a certain way. For an example, see **Customize attribute files, part mapping and user-defined attributes (UDAs)**.

Rebar attributes tab

Use the **Rebar attributes** tab to define the user-defined attributes (UDAs) for rebars.

Option	Description
Threaded length	The value is written to the reinforcing bar UDA. The fields are used for checking which bars have threads and what are the thread values so that they can be shown in drawings and reports.
Extra fabrication length	This value is written to the reinforcing bar UDA. This value does not affect the length of the reinforcing bar. You need to add this value in your drawings and reports to get the correct length if required.
Method **Type** **Product** **Code** **Thread type** **Free distance 1** **Free text 2**	UDAs written to the reinforcing bars. You can set the reinforcing bar end method and the coupler type, and add a product name and a code for reporting purposes. The used attribute name depends on which end of the reinforcing bar the coupler was created.

NOTE The above user-defined attributes affect numbering. Reinforcing bars with different attribute values get different part marks.

Coupler attributes tab

Use the **Coupler attributes** tab to define user-defined attributes (UDAs) for couplers or end anchor parts. For each UDA, you can select to inherit the value from the rebar or the cast unit. To enter a value, select the inherit option **None**.

Note that if the cast unit does not have any value for that particular UDA, the value is taken from the main part of the cast unit. By default, you can enter a **Comment** and values for **User field 1 ... 4** UDAs. You can customize the set of UDAs by modifying the UDA fields in the `RebarCouplers.Udas.dat` file, located in the `..\Environments\common\system` folder.

Split rebar and add coupler

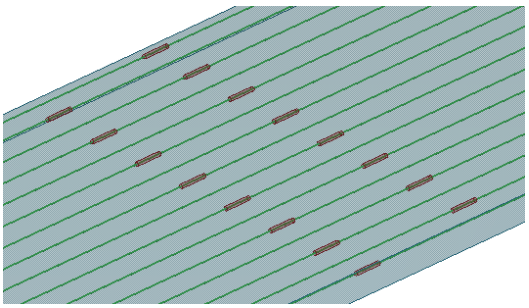
Split rebar and add coupler splits a reinforcing bar group or rebar set, and adds couplers in relation to the direction of the picked points.

Objects created

- Circular hollow parts

  The parts connect the two reinforcing bar groups or rebar sets.

Use for

Situation	Description
	Staggered couplers with split reinforcing bars.

Selection order

1. Double-click **Split rebar and add coupler** to open the component properties.
2. Click **Split rebar with coupler**.
3. Select the reinforcing bar, bar group, or rebar set to be split.
4. Pick the first split point.
5. Pick the second split point.

   This point defines the line at which the bars are split and couplers are inserted.

6. Pick the third point.

   This point defines the side of the primary bars. This is needed if the primary and secondary bars have different properties.

7. Repeat the steps 3 - 6, or press **Esc** to cancel picking.

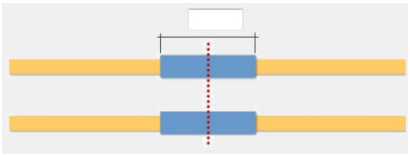
Parameters tab

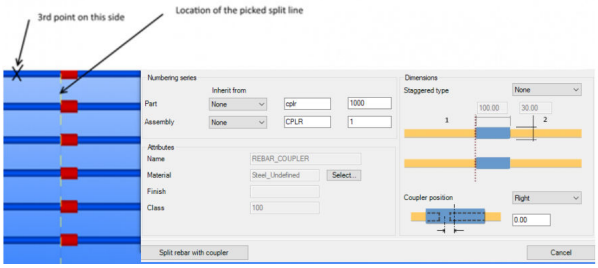
Use the **Parameters** tab to define the coupler properties.

Option	Description
Use manually entered values	Enter values for the coupler properties.
Use auto attribute file	The values in the auto attribute file override the coupler component values. Typically the auto attributes file contains one or more attribute values to be used with certain bar sizes. Go to Tekla Warehouse to get auto attribute files for your project. For more information on the structure of the auto attribute files, see **Customize attribute files, part mapping and user-defined attributes (UDAs)**.

Option	Description
Use custom component	Select **Yes** to create the coupler as a custom part. Select **No** to create the coupler as a normal part.
Name	Type the name of the custom component, or select it from the **Applications & components** catalog. Ensure that the selected component is a custom part.
Attribute	Type the name of the attribute file of the custom component. The coupler custom part is created using the saved attributes given here.
Input points	The order of the start and the end points of the custom component.
Add to rebar assembly	To add the coupler as a sub-assembly to the rebar assembly, select either **Yes, to primary bar** or **Yes, to secondary bar**. These options

Option	Description
	will create new rebar assemblies for the new bars created by the split command. The default value is **No**.

Option	Description
Numbering series	Prefix and a start number for the coupler part position number. In the **Inherit from** list, select which numbering settings are used for couplers. • With **None**, enter the part and assembly prefix and start numbers. • With **Rebar**, the numbering settings of the primary bars are used. • With **Cast unit**, the cast unit numbering settings are used.
Attributes	Name, diameter, material, finish and class of the coupler parts.
Staggered type	• **None** Couplers are placed along the two picked points. • **Left** Couplers are placed to the left side of the two picked points. • **Right** Couplers are placed to the right side of the two picked points. • **Middle** Couplers are placed on both sides of the two picked points.
	The overall end-to-end length of the coupler. If you use a custom part, this is the length between the start point and end the point of the custom part.
Coupler position	Location of the couplers in relation to the selected split line. If you select the **Left** or the **Right** option, the actual left or right depends on the third input point which defines the side of the first bar.

Option	Description
	This option is available only when there is no staggering. Example: 

When you create the couplers as custom parts, the properties in the **Numbering series** and **Attributes** sections can be filled from the custom part settings if you have named the properties in a certain way. For an example, see **Customize attribute files, part mapping and user-defined attributes (UDAs)**.

Rebar attributes tab

Use the **Rebar attributes** tab to define the user-defined attributes (UDAs) for rebars.

Option	Description
Threaded length	The value is written to the reinforcing bar UDA. The fields are used for checking which bars have threads and what are the thread values so that they can be shown in drawings and reports.
Extra fabrication length	This value is written to the reinforcing bar UDA. This value does not affect the length of the reinforcing bar. You need to add this value in your drawings and reports to get the correct length if required.
Method **Type** **Product** **Code** **Thread type** **Free distance 1** **Free text 2**	UDAs written to the reinforcing bars. You can set the reinforcing bar end method and the coupler type, and add a product name and a code for reporting purposes. The used attribute name depends on which end of the reinforcing bar the coupler was created.

NOTE The above user-defined attributes affect numbering. Reinforcing bars with different attribute values get different part marks.

Coupler attributes tab

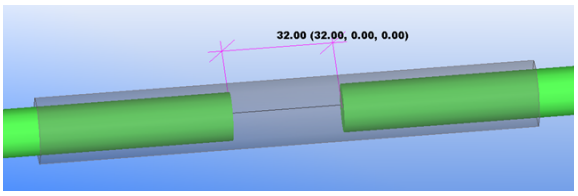
Use the **Coupler attributes** tab to define user-defined attributes (UDAs) for couplers or end anchor parts. For each UDA, you can select to inherit the value from the rebar or the cast unit. To enter a value, select the inherit option **None**.

Note that if the cast unit does not have any value for that particular UDA, the value is taken from the main part of the cast unit. By default, you can enter a **Comment** and values for **User field 1 ... 4** UDAs and **Layer**. You can customize the set of UDAs by modifying the UDA fields in the `RebarCouplers.Udas.dat` file, located in the `..\Environments\common\system` folder.

Rebar end trimming

Rebar end trimming adjusts the space between two reinforcing bar ends.

Use for

Situation	Description
	Space between reinforcing bar ends.

Selection order

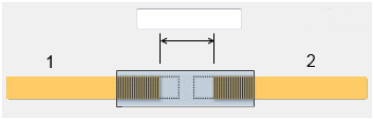
1. Select the couplers you want to modify.
2. Modify the properties.
3. Click **Apply to selected couplers**.

OR

1. Modify the properties.
2. Click **Select rebars**.
3. Select the reinforcing bars whose ends are adjusted according to the value set for the gap.

Rebar coupler/Rebar end anchor tab

Option	Description
Get free gap from auto attributes file	Select **Yes** if you want to use the gap defined in the **Auto attribute file** for the couplers. When

Option	Description
	you select **Yes**, only the coupler components that have the **Use auto attribute file** in use are adjusted. See **Customize attribute files, part mapping and user-defined attributes (UDAs)**. Select **No** if you want to manually enter the free gap.
	The gap when **Get free gap from auto attributes file** is set to **No**.
Trim both bars	Select which of the reinforcing bars is modified.
Apply to selected couplers	Select the couplers you want to modify and click the **Apply to selected couplers** button to adjust the bars according to the value set for the gap.
Select rebars	Click the **Select rebars** button and select the reinforcing bars whose ends you want to adjust according to the value set for the gap. The reinforcing bar ends need to be close to each other.

Update rebar attributes

Use **Update rebar attributes** to manage the user-defined attributes (UDAs) of the couplers and the end anchor parts created by the **Rebar Coupler and Anchor Tools**. With **Update rebar attributes** you can check the current values of the selected reinforcing bars or all reinforcing bars.

If you have added coupler or end anchor components to reinforcing bars, the UDA values are controlled by the coupler or the end anchor components and their properties. If you delete the coupler or the end anchor component, the UDAs defined by those components are not cleared automatically. Use **Update rebar attributes** to clear the obsolete attribute values.

Option	Description
Selected	The values of all coupler and end anchor UDAs of the selected reinforcing bars in the model.
All	The values of all coupler and end anchor UDAs of all reinforcing bars in the model.

Option	Description
Update	Deletes the values of all coupler and end anchor UDAs of all reinforcing bars on the selected rows. Only the obsolete coupler UDAs of the reinforcing bar are deleted. You can select multiple rows by holding down **Ctrl** or **Shift**.
Update (phase only)	Updates the phase values of the coupler and end anchor parts so that the phase values are the same as the phase values of the related reinforcing bars.
Show only rebar with attributes	Select this check box if you want to show only reinforcing bars that have values in their coupler or end anchor UDAs. After you have selected the check box, click **Selected** or **All** to refresh the table.

Customize attribute files, part mapping and user-defined attributes (UDAs)

Auto attribute files

The attribute table files are text files that can locate in any of the system folders, or in a model folder. You can have as many attribute table files as you need. There are different attribute tables for end anchors (one input reinforcing bar) and coupler components (two input reinforcing bars). The extensions for attribute table files are

- `.couplers.csv` for **Rebar coupler** and **Split rebar and add coupler** components
- `.anchors.csv` for **Rebar end anchor** component.

The attribute table contains a header line, including the column names and one or more table rows containing the attribute values. Columns are either selector columns or attribute columns.

The selector column names are `Primary.Size`, `Primary.Grade`, `Secondary.Size`, and `Secondary.Grade`.

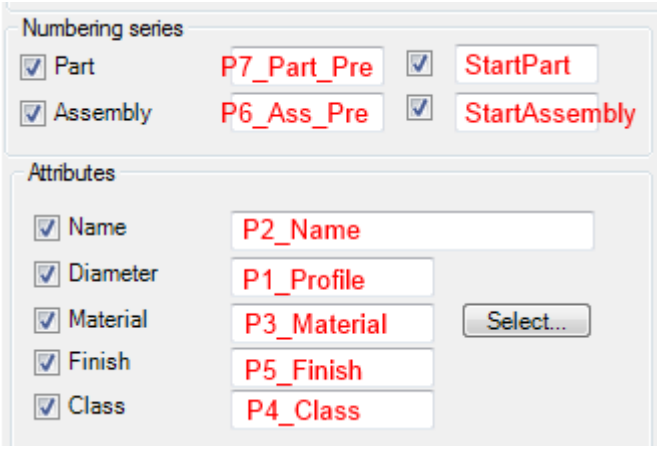
The attribute columns contain the attribute value that is the name given in the header row. The component attribute values given in the table row are used whenever the component input (primary + secondary for **Rebar coupler** and secondary for **Rebar end anchor**) matches with the selector values.

Go to Tekla Warehouse to get auto attribute files for your project.

Custom part mapping

When you create the coupler or the end anchors as custom parts, the properties in the **Numbering series** and **Attributes** sections can be filled from the custom part settings if you have named the properties in a certain

way. The following example shows the mapping between the **Numbering series** and **Attributes** properties and the custom part properties.



User-defined attributes (UDAs)

You can customize the content of the **Attributes** tab and the UDAs in the **Rebar Coupler and Anchor Tools** components.

The UDAs are defined in a text file named `RebarCoupler.Udas.dat`, located in the `...\ProgramData\Trimble\Tekla Structures\<version>\Environments\Common\system` folder. The file can also be placed in some other system folder or in a firm folder. Note that the file is not read from the model folder.

The `RebarCoupler.Udas.dat` file only defines which UDAs are shown on the **Attributes** tab. If you modify the file, include only existing UDAs to the file. If you want to create new UDAs, ensure that you define them properly.

NOTE The UDAs affect numbering if the UDAs' special numbering flag (`special_flag`) is set to `yes` in the `objects_couplers.inp` file. Reinforcing bars with different UDA values will then get different part marks.

Slab bars (18)

Slab bars creates reinforcement for a concrete slab.

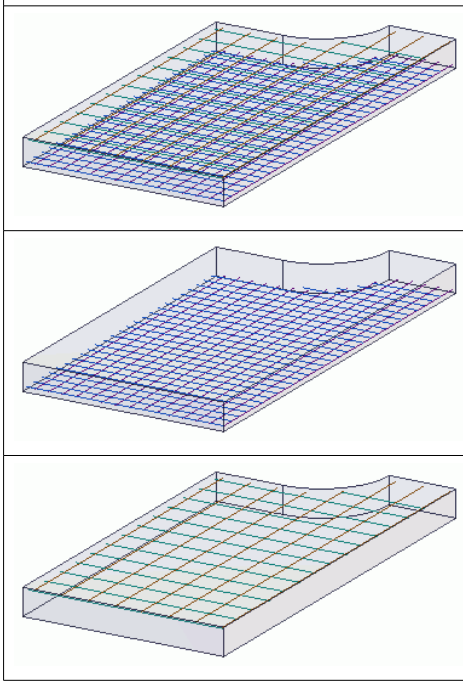
NOTE Alternatively, you can use the **Mesh Bars** and the **Mesh Bars by Area** components to create reinforcement to concrete slabs or walls.

See Mesh Bars / Mesh Bars by Area (page 3863).

Bars created

- Primary slab bars
- Crossing slab bars

Use for

Situation	More information
	**Slab bars (18)** creates reinforcement for the bottom or top surface of the slab, or for both.

Before you start

- Create the concrete slab.
- Calculate the required area of reinforcement.

Selection order

1. select the concrete slab.

Picture tab

Use the **Picture** tab to define concrete cover thickness, which bars to create (bottom/top/both), primary bar direction, and which sides are regarded as top and bottom sides.

Field	Description
Create bars	Defines on which surface(s) to create bars. The options are: • **Both sides** • **Bottom side** • **Top side**
Primary bar direction	The direction of primary bars. Use to change bar direction. The options are: • **Use slab x direction**

Field	Description
	• **Use slab y direction** • **Use global x direction** • **Use global y direction**
Up direction	Defines which sides are regarded as the top and bottom sides of the slab.

Bottom bars/Top bars tab
Use the **Bottom bars** and **Top bars** tabs to define grade, size, spacing, and generation type of bottom/top bars, concrete cover thickness on slab sides, crossing bar location in relation to primary bars, end hooks and bend lengths.

Field	Description
Bar generation type	Defines whether Tekla Structures treats the bars as a group or a mesh.
Cross bar location	Defines whether the crossing bars are located above or below the primary bars.
End conditions	The hook angles at bar ends. The options are **Straight**, 90, 135, and 180 degrees.
Bend lengths	The bend lengths at bar ends.

Attributes tab
Use the **Attributes** tab to define numbering properties, name, and class of bars.

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

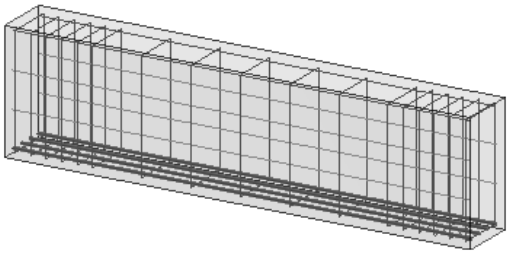
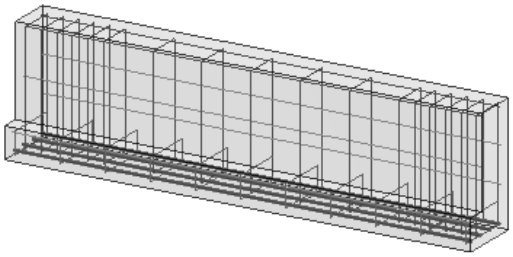
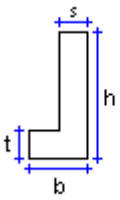
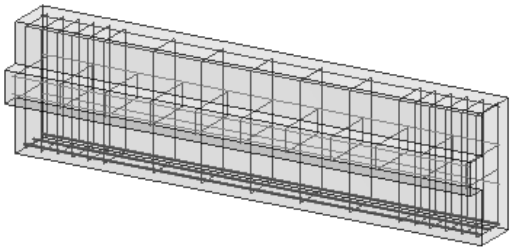
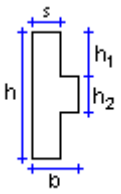
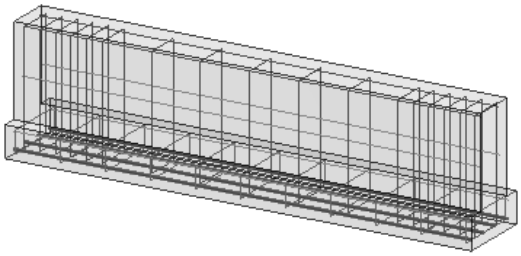
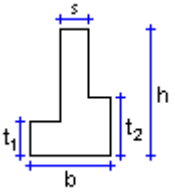
Beam reinforcement (63)
Beam reinforcement (63) creates reinforcement for a concrete beam.

Bars created

- Longitudinal main bars

- Corner and side bars
- Main stirrups
- Stirrups for one or two ledges

Use for

Situation	More information
	Rectangular beams
	L-shaped beams (RCL profiles) 
	RCX profiles 
	Inverted T-beams (RCDL profiles) 

Before you start

- Create the concrete beam.
- Calculate the required area of reinforcement.

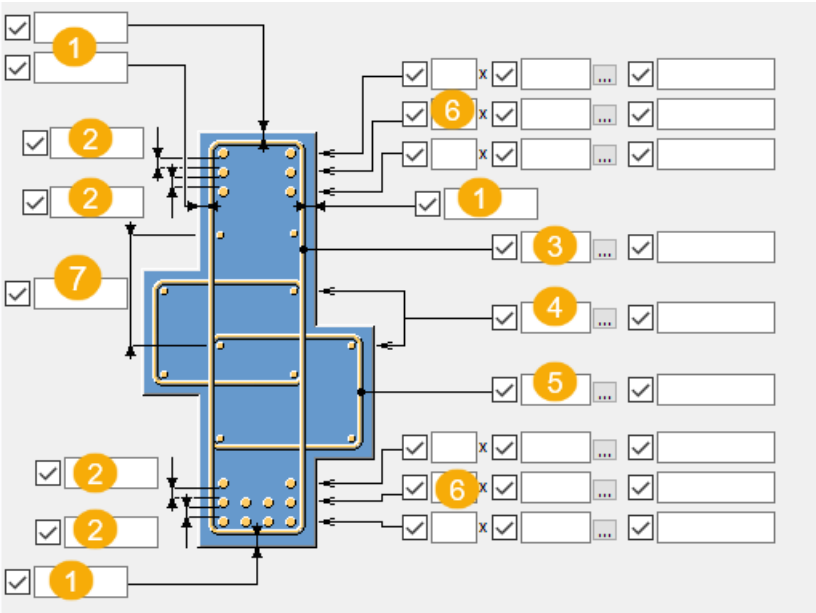
Selection order

1. Select the concrete beam.

   The reinforcement is created automatically when you select the beam.

Parameters tab

Use the **Parameters** tab to define the concrete cover thickness, number, size, and location of the bars, and stirrup size.



	Description
1	Cover thickness
2	Distance between bars
3	Main stirrup size
4	Corner and side bar size
5	Ledge stirrup size
6	Main bars number and sizes
7	Sidebar spacing, based on maximum target spacing

Bottom main bars

Select how the bottom main bars are positioned: inside the main stirrups or distributed to the ledges. You can select the position when the bottom main bars have the same height as at least one of the ledges.

Placing of top and bottom bars

Select the horizontal placing for the top and bottom bars. Select whether the bars are placed starting from the corner or at equal spaces.

Option	Description
	At equal spaces
	From the corner

Bending direction

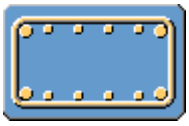
Option	Description
	Type 1
	Type 2
	Type 3
	Type 4

Stirrup shape

Option	Description
	90-degree

Option	Description
	135-degree
	Overlapped

Additional links

Option	Description
	No additional links.
	Create additional internal links.
	Create additional links.

Select whether links are created always or depending on the profile size from the **Create links** list.

If you select **If profile size >**, enter the minimum profile length.

Side bars

Select whether the side bars are created as one group or as separate rebar groups.

Hooks tab

Use the **Hooks** tab to create hooks and define their properties.

Define the hook type and bar end cover separately for the start and end of the rows.

Option	Description
	No hook
	Standard 90-degree hook
	Standard 135-degree hook
	Standard 180-degree hook
	Custom hook

If you select a standard hook, the **Angle**, **Radius**, and **Length** use predefined dimensions.

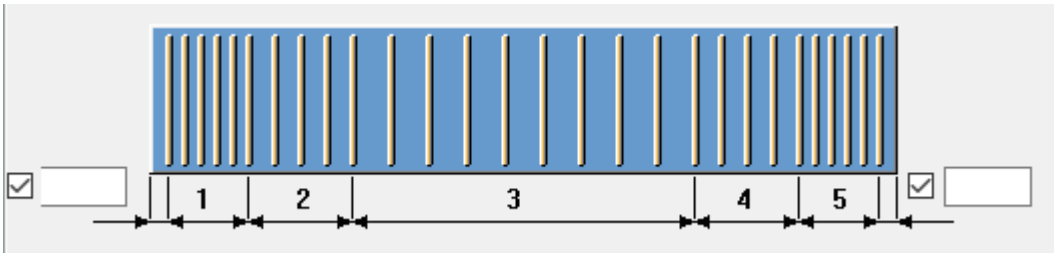
Option	Description
Angle	Enter a value between -180 and +180 degrees.
Radius	Enter the internal bending radius of the hook.
Length	Enter the length of the straight part.

Stirrup spacing tab

Use the **Stirrup spacing** tab to define whether stirrups are created as one group or as separate groups, and to define stirrup spacing.

Stirrup spacing

Define the distances from the part ends to the first stirrups.



Option	Description
N	Number of stirrups.
Space	Define the spacing of stirrups. Use a space to separate spacing values. Enter a value for each space between the stirrups. For example, if there are 3 stirrups, enter 2 values.

Main stirrup spacing type and Ledge Stirrup Spacing Type

Define how the main stirrups and ledge stirrups are distributed along the length of the beam.

Option	Description
Target	Tekla Structures creates equal spaces between the bars, aiming the spacing value as closely as possible to the value you specify. This is the default option.
Exact	The first and last space of a zone adjust themselves to even out bar

Option	Description
	distribution. The spaces in the middle of each zone are exactly the size you specify.

Advanced tab
Use the **Advanced** tab to define the naming and numbering properties of bars and stirrups.

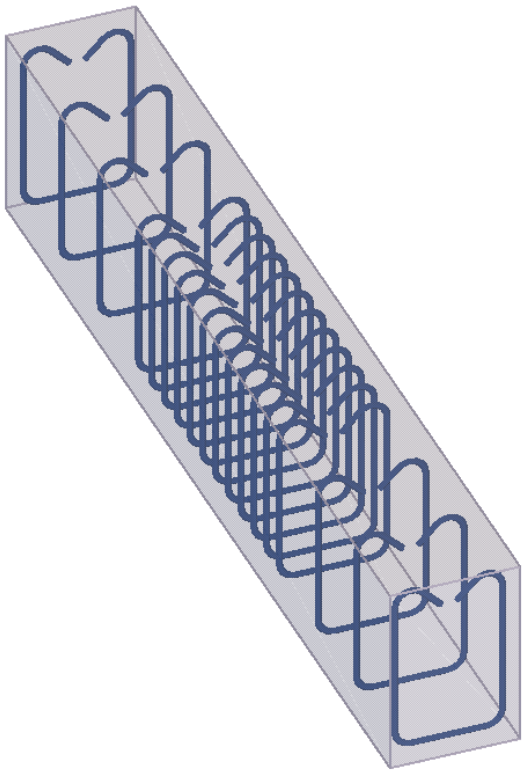
Option	Description
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Prefix	Prefix for the bar position number.
Start number	Start number for the bar position number.

Stirrup reinforcement (67)
Stirrup reinforcement (67) creates stirrup reinforcement for a beam, a panel, or a rectangular slab.

Reinforcement created

- Main stirrups
- Hooks (Optional)

Use for

Situation	More information
	A concrete beam with main stirrups.

Before you start

- Create the concrete part.
- Calculate the required area of reinforcement.

Parameters tab

Use the **Parameters** tab to define stirrup properties.

Reinforcing bar

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Size	Diameter of the reinforcing bar.

Option	Description
Grade	Strength of the steel used in the reinforcing bars.
Bending radius	Internal radius of the bends in the bar. You can enter a separate value for each bar bend. Separate the values with spaces. Bending radius complies with the design code you are using. Main bars, stirrups, ties, and hooks usually have their own minimum internal bending radii, which are proportional to the diameter of the reinforcing bar. The actual bending radius is normally chosen to suit the size of the mandrels on the bar-bending machine.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Hook type

Define the hook type separately for the start and end of the part. The options are:

Option	Description
	No hook
	Standard 90-degree hook
	Standard 135-degree hook
	Standard 180-degree hook
	Custom hook

If you select a standard hook, the **Angle**, **Radius**, and **Length** use predefined dimensions.

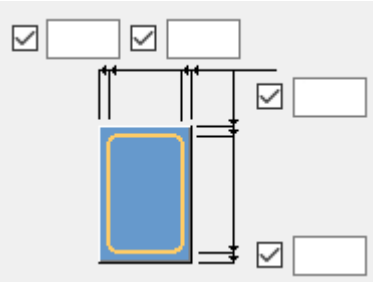
Option	Description
Angle	Enter a value between -180 and +180 degrees.
Radius	Enter the internal bending radius of the hook.

Option	Description
Length	Enter the length of the straight part.
Stirrup shape	Select the shape of the stirrups. The options are **Open**, **Closed**, and **Spiral**.
Number of rounds, N*	If you select the stirrup shape **Spiral**, enter the number of rounds the spiraled stirrup has.

Cover thickness

Define whether the concrete cover thickness is same or different on each side from **Concrete cover** dropdown list.

If you select a different concrete cover for each side, enter the separate cover thicknesses.



Rotation

Rotate the polygon of the rebar shape from the **First stirrup** and the **Second stirrup**. You can rotate the polygon either at each even or odd position.

Stirrup spacing tab

Use the **Stirrup spacing** tab to define the spacing of stirrups.

You can define six zones for the distribution.

Option	Description
Symmetric	Define whether the same spacing and bundling properties are used at the start and end of the part.
N	Number of stirrups.
Space	Define the spacing of stirrups. Use a space to separate spacing values. Enter a value for each space between the stirrups. For example, if there are 3 stirrups, enter 2 values.

Option	Description
Bundling	Select whether to have single or double main stirrups. Single stirrups: Bundled stirrups:
Min hole size	Define a size limit for the holes.
Stirrup spacing type	**Target** Tekla Structures creates equal spaces between the bars, aiming the spacing value as closely as possible to the value you specify. This is the default option.
	Exact The first and last space of a zone adjust themselves to even out bar distribution. The spaces in the middle of each zone are exactly the size you specify.

Holes and recesses tab

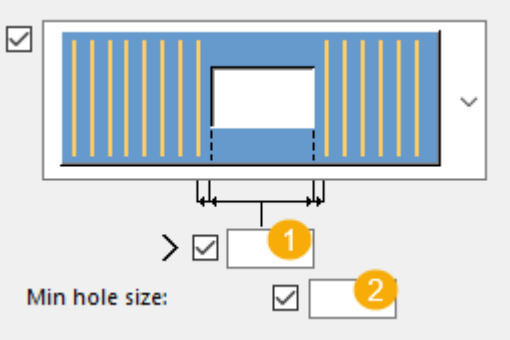
Use the **Holes and recesses** tab to define how reinforcement is created around openings.

Bar behavior at holes and recesses

Select how the bars next to holes and recesses are handled.

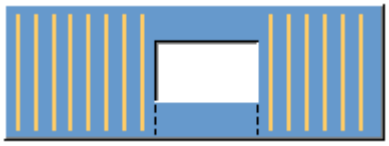
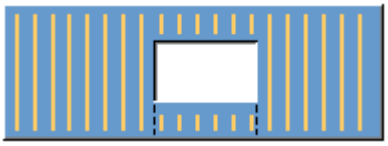
Option	Description
Cut bars	Select whether the bars are cut and which cuts are ignored. You can select to ignore cuts by part name or class. If you ignore cuts by name, define the name. If you ignore cuts by class, define the class.

Cover thickness



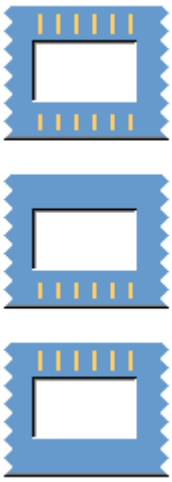
	Description
1	Define the cover thickness of the opening to control where the first full height stirrup is placed.
2	Define the minimum hole size.

Reinforcement around opening

Option	Description
	No reinforcement around the opening.
	Reinforcement is created around the opening.

Reinforcement position

If you create reinforcement around the opening, select how the reinforcement is positioned.



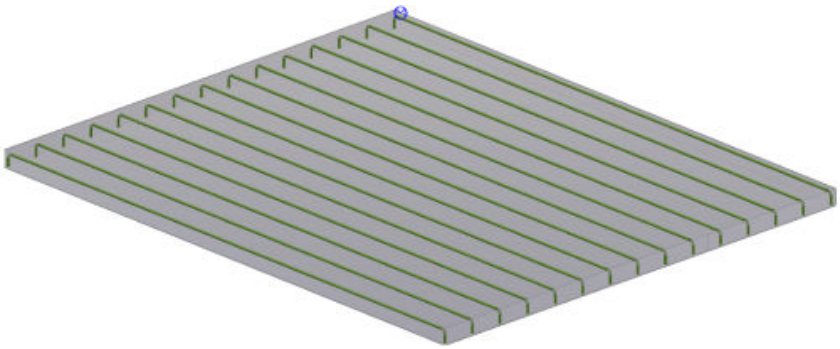
Longitudinal reinforcement (70)

Longitudinal reinforcement (70) creates longitudinal reinforcing bars to concrete parts.

Objects created

- Main bars
- Hooks (Optional)

Use for

Situation	Description
	Concrete slab with longitudinal main bars and standard hooks.

Before you start

- Create the concrete part.
- Calculate the required area of reinforcement.

Selection order

1. Select the concrete part.

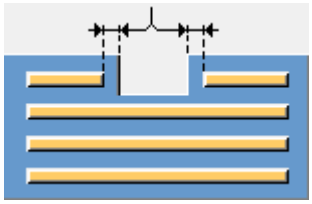
Parameters tab
Use the **Parameters** tab to define the longitudinal bars properties and spacing.

Properties

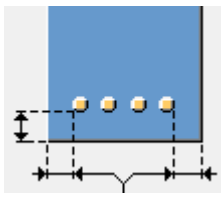
Option	Description
Prefix	Prefix for the part position number.
Start No	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Size	Diameter of the reinforcing bar.
Grade	Strength of the steel used in the reinforcing bars.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Concrete cover thickness

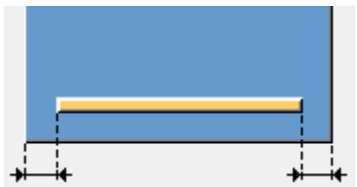
If the part has a cut, enter the concrete cover thicknesses at cut positions.



Enter the concrete cover thicknesses on the plane and from the plane.



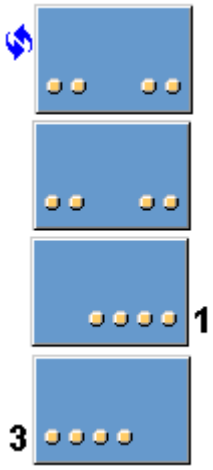
Enter the concrete cover thickness at the start and end of the bars.



Spacing

Option	Description
Spacing	Define the spacing of reinforcement bars. Use a space to separate spacing values. Enter a value for each space between the bars. For example, if there are 3 bars, enter 2 values.
Number of bars	Enter the number of reinforcing bars.

Select the location of reinforcing bars from the list. The default is that the bars are on both sides.



Hooks tab

Use the **Hooks** tab to create hooks and define their properties.

Hook type

Define the hook type separately for the start and end of the part. The options are:

Option	Description
	No hook
	Standard 90-degree hook
	Standard 135-degree hook
	Standard 180-degree hook
	Custom hook

If you select a standard hook, the **Angle**, **Radius**, and **Length** use predefined dimensions.

Option	Description
Angle	Enter a value between -180 and +180 degrees.
Radius	Enter the internal bending radius of the hook.
Length	Enter the length of the straight part.

Select the direction of hooks from the **Direction** list.

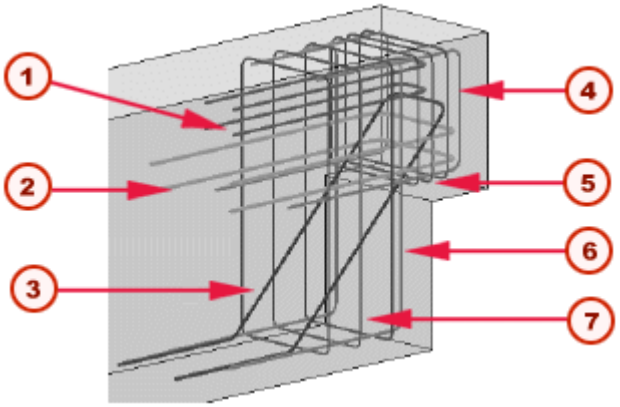
Select to create the hooks for split rebars from the **Hooks for split rebars** list.

Beam end reinforcement (79)

Beam end reinforcement (79) creates reinforcement for the end of a concrete beam or strip footing.

Bars created

- Horizontal U-shaped bars (types 1 and 2)
- Vertical U-shaped bars (types 3A and 3B)
- Oblique bar (type 4)
- Stirrups (types 5A and 5B)



	Description
1	Horizontal U bars (type **2**)
2	Horizontal U bars (type **1**)
3	Oblique bar (type **4**)
4	Vertical U bars (type **3A**)
5	Stirrups (type **5A**)
6	Vertical U bars (type **3B**)

	Description
7	Stirrups (type **5B**)

Use for

Situation	More information
Standard beams	Use bars 3A and 5A for the beam end.
Dapped beams	
Beams with anchor bolt hole in the notched area	
Beams and strip footings that have a rectangular cross section	

Do not use for

Parts that have irregular cross sections.

Before you start

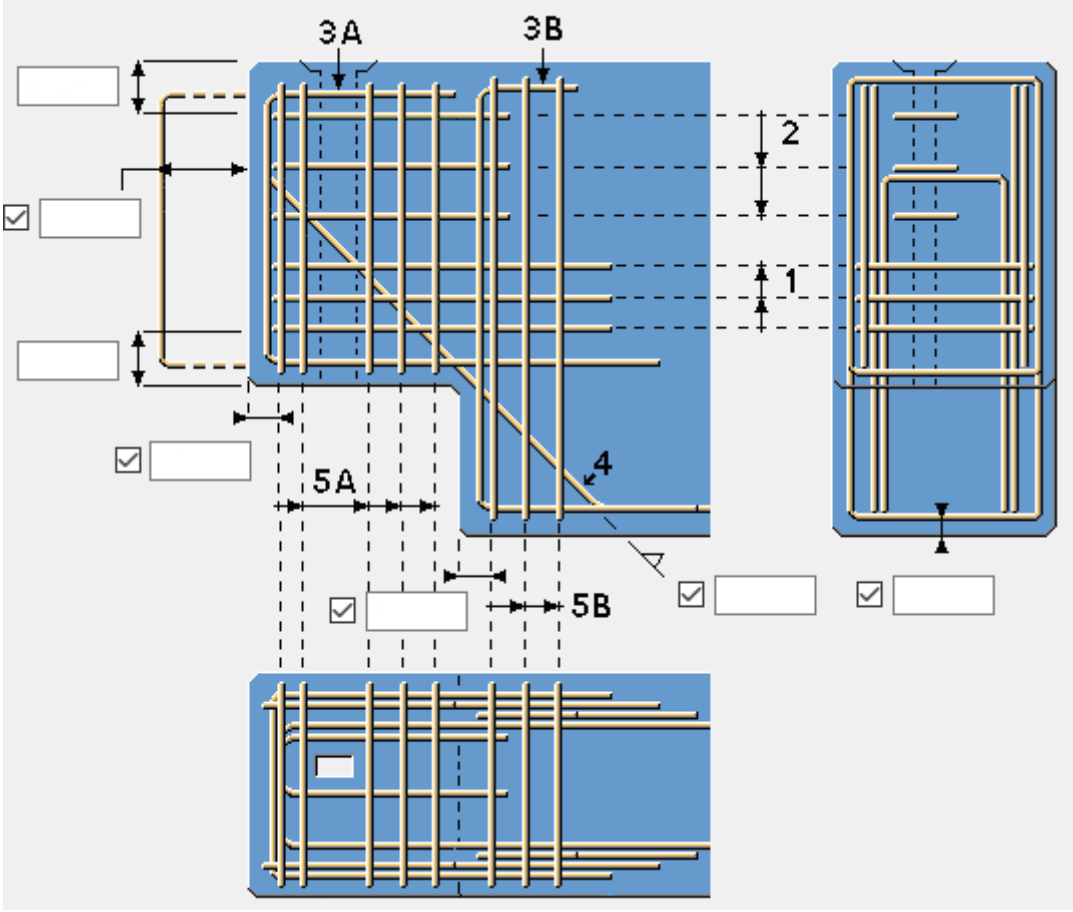
- Create the concrete beam or strip footing.
- Calculate the required area of reinforcement.

Picking order

1. Select concrete beam or strip footing.
2. Pick position.

Picture tab

Use the **Picture** tab to define concrete cover thickness, distances from the concrete surface to the bars, and angle of bar 4.

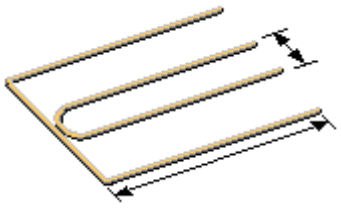


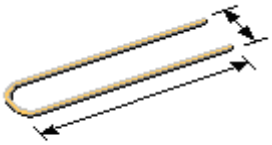
Bars tab

Use the **Bars** tab to define which bars to create, bar dimensions, and to slice bars 3A and 3B.

Horizontal U bars 1 and 2

Use the following options to create bars in the lower area of the beam end, in the horizontal planes (bar type 1):

Option	Description
	Two bars on each plane. One in the middle of the beam end, the other extending to the sides of the beam.

Option	Description
	One bar on each plane, extending to the sides of the beam.
	One bar on each plane, in the middle of the beam end.
	Two identical, overlapping bars on each plane.

To create bars in the upper area of the beam end, around a hole, enter dimensions for bar 2.

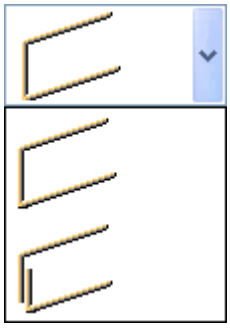
Vertical U bars 3A and 3B

To create vertical U-shaped bars, enter dimensions for:

- Bar 3A: for the notched area of the beam.
- Bar 3B: for the higher part of the beam.

Splicing bars 3

You can create vertical U bars (bar type 3) of two bars joined with a splice. To do this, use the following list box on the **Bars** tab:



If you choose to splice bars, you can select the splice type:

Option	Description
Lap up	Creates a lap splice above the horizontal center line of the beam end.
Lap down	Creates a lap splice below the horizontal center line of the beam end.
Lap both	Creates a lap splice centered to the horizontal center line of the beam end.
Coupler	Creates a coupler.
Weld joint	Creates a welded joint.

For lap splices, you can define the lap length **L** and whether the bars are on top of each other or parallel to each other.

For all splice types, you can define the offset of the splice center point from the horizontal center line of the beam end.

Stirrups 5A and 5B

To create stirrups for beam ends, enter dimensions for:

- Bar 5A: for the notched area of the beam.
- Bar 5B: for the higher part of the beam.

Groups tab

Use the **Groups** tab to define grouping properties of bars.

Enter the number and spacing of bars in each group of bar types. If the spacing varies, enter each value individually.

Attributes tab

Use the **Attributes** tab to define bar properties.

Option	Description
Grade	Strength of the steel used in the reinforcing bars.
Size	Diameter of the reinforcing bar.
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Define the name for the reinforcing bars. Tekla Structures uses the name in drawings and reports.

Option	Description
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

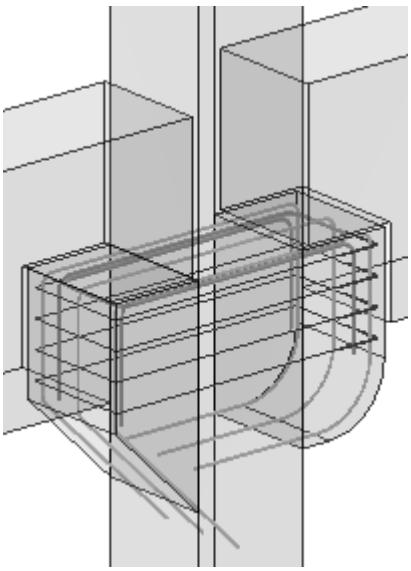
Corbel reinforcement (81)

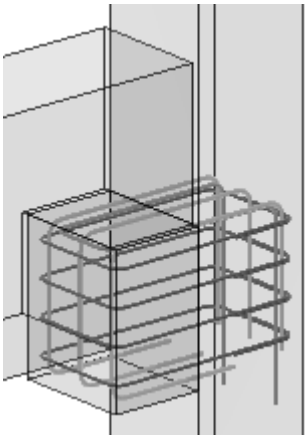
Corbel reinforcement (81) creates reinforcement for one or two corbels in a concrete column. The two corbels must have the same top level, thickness, and horizontal location.

Objects created

- Main bars
- Stirrups
- Additional bars

Use for

Situation	Description
	Two corbels, beveled and rounded, with the same top level, thickness, and horizontal location. Two additional bars crossing each other.

Situation	Description
	One straight corbel. One additional bar.

Limitations

Do not use **Corbel reinforcement (81)** for two very different corbels.

Before you start

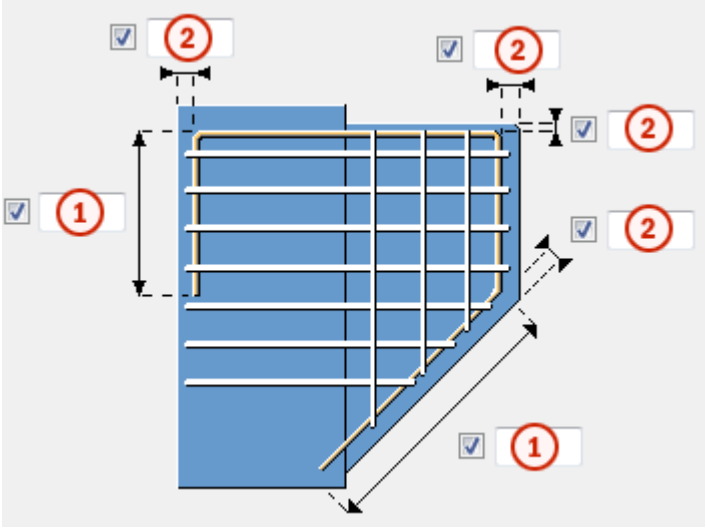
- Create the concrete column and beams.
- Create the corbels. Use **Corbel connection (14)** or a seating connection to create the corbels.

Selection order

1. Select the main part (concrete column).
2. Select the secondary parts (one or more corbels).
3. Click the middle mouse button to create the reinforcement.

Main bars tab

Use the **Main bars** tab to control the main bar length, concrete cover thickness, reinforcing bar spacing, and reinforcing bar properties.



	Description
1	Main bar length.
2	Distance from the concrete surface to the main bar.

Option	Description
Grade	Strength of the steel used in the reinforcing bars.
Size	Diameter of the reinforcing bar.
Name	Define a name for the main bars. Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.

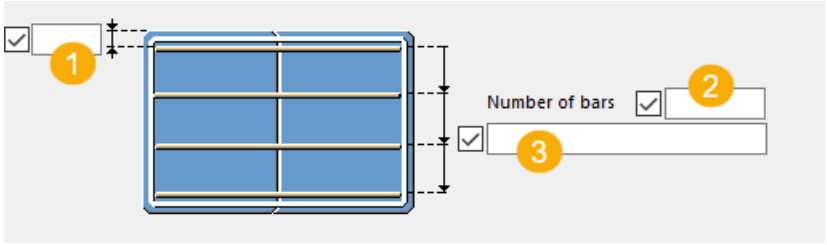
Follow shape

This setting also affects the transverse stirrups.

Option	Description
	Inclined The last rebar segment follows the inclined corbel edge.

Option	Description
	Horizontal The last rebar segment is created horizontally.

Cover thickness



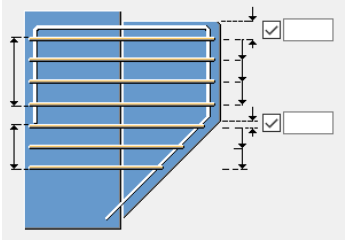
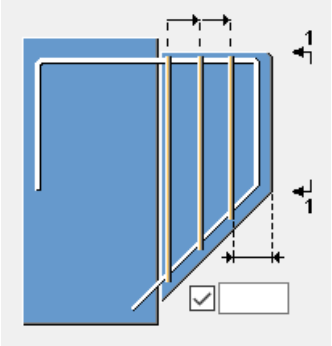
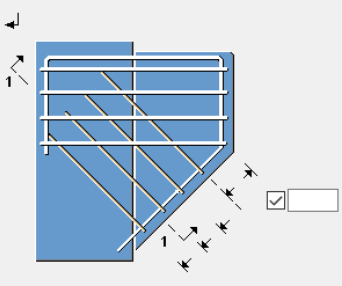
	Description
1	Concrete cover thickness.
2	Number of bars in the reinforcement.
3	Space between the reinforcing bars.

Stirrups/Transverse stirrups/Diagonal stirrups tab

Use the **Stirrups**, **Transverse stirrups** and **Diagonal stirrups** tabs to control the stirrup creation, stirrup type, concrete cover thickness, reinforcing bar spacing and properties.

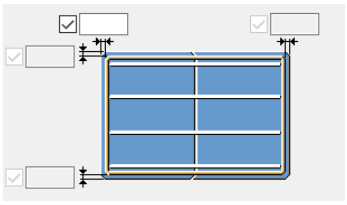
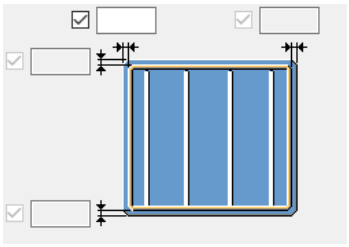
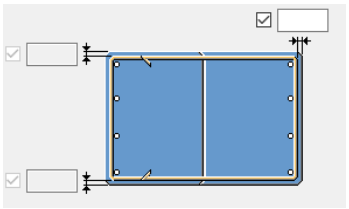
Stirrups

Define the distance from the concrete surface to the stirrups.

Option	Option	Option
		

Concrete cover thickness

Define the concrete cover thickness. You can select that the concrete cover is the same at each side.

Option	Option	Option
		

Option	Description
Number of bars	Define the number of bars in the reinforcement.
Spacing values	Define the space between the reinforcing bars.
Transverse stirrups in column	Select whether the column is reinforced with transverse stirrups.
Name	Define a name for the stirrups. Tekla Structures uses the name in drawings and reports.
Size	Diameter of the reinforcing bar.
Grade	Strength of the steel used in the reinforcing bars.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.

Stirrup type

Option	Description
	The stirrup consists of a bent single reinforcing bar. Define the hook length. The default values (for the standard 90 degree stirrup hook) are read from the `rebar_database.inp` file. Select a suitable overlap position for this stirrup type.
	The stirrup consist of two overlapping U bars that face each other. Define minimum and maximum overlapping length.

Option	Description
	The stirrup consists of two overlapping U bars. Define minimum and maximum overlapping length.
	The transverse stirrup is one open U-bar.

Additional bars tab

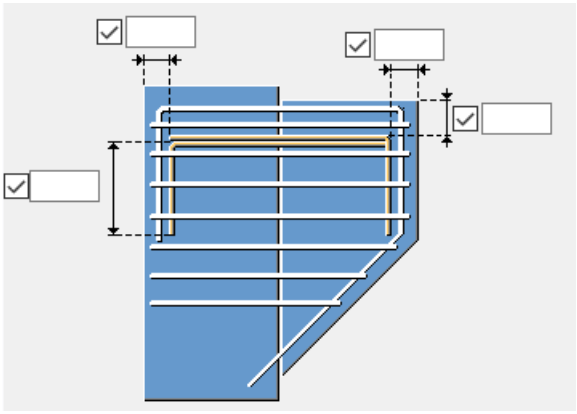
Use the **Additional bars** tab to control the creation of additional bars in corbels, concrete cover thickness, and additional bar properties.

Additional bars

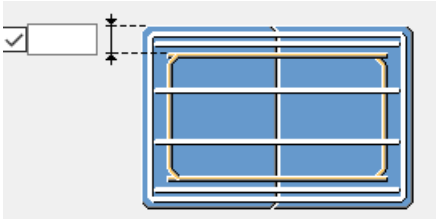
Option	Description
	No additional bars.
	One additional bar.
	Two additional bars parallel to each other.
	Two additional bars crossing each other.
	Two pairs of additional bars crossing each other.

Stirrups and cover thickness

Define the distance from the concrete surface to the stirrups and the additional bar dimension.



Define the concrete cover thickness.



Option	Description
Grade	Strength of the steel used in the reinforcing bars.
Size	Diameter of the reinforcing bar.
Name	Define a name for the additional bars. Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.

Rebar assembly tab

Use the **Rebar assembly** tab to add the created reinforcement as a rebar assembly to the cast units.

Create as rebar assembly

You can add the created reinforcement as a rebar assembly to the cast units with a predefined assembly type, name, prefix, and start number.

Option	Description
Create as rebar assembly	Select **Yes** to create all reinforcement as one rebar assembly, and to include it to the cast unit of the input part.
Add to existing rebar assembly	• **Do not add**: New rebar is added as a rebar assembly to the cast unit. • **As single bars**: Add all new rebars directly into an existing rebar assembly as rebars. • **As sub-assembly**: Add all new rebars to their own new rebar assembly, which is then included as a sub-assembly to the main rebar assembly.
Rebar assembly type	Select the rebar assembly type. If you do not select the type, the default value of the rebar assembly is used.
Name, **Profile**, **Start number**	Define the name, profile, and start number. If you do not define these, the default values of the rebar assembly are used.

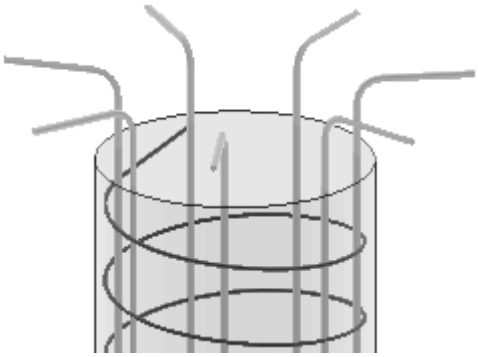
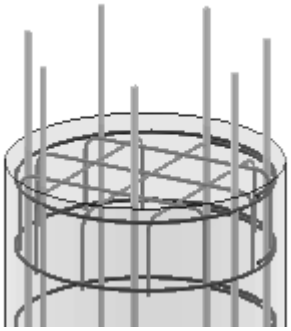
Round column reinforcement (82)

Round column reinforcement (82) creates reinforcement for a concrete column that has a round cross section.

Objects created

- Longitudinal main bars (6)
- Stirrups
- Column end reinforcement

Use for

Situation	Description
	Round concrete column with main bars bent outside the column. Continuous spiral stirrup.
	Round concrete column with straight main bars protruding from the column, or entirely inside the column. Separate stirrups. Top of the column is reinforced.

Limitations

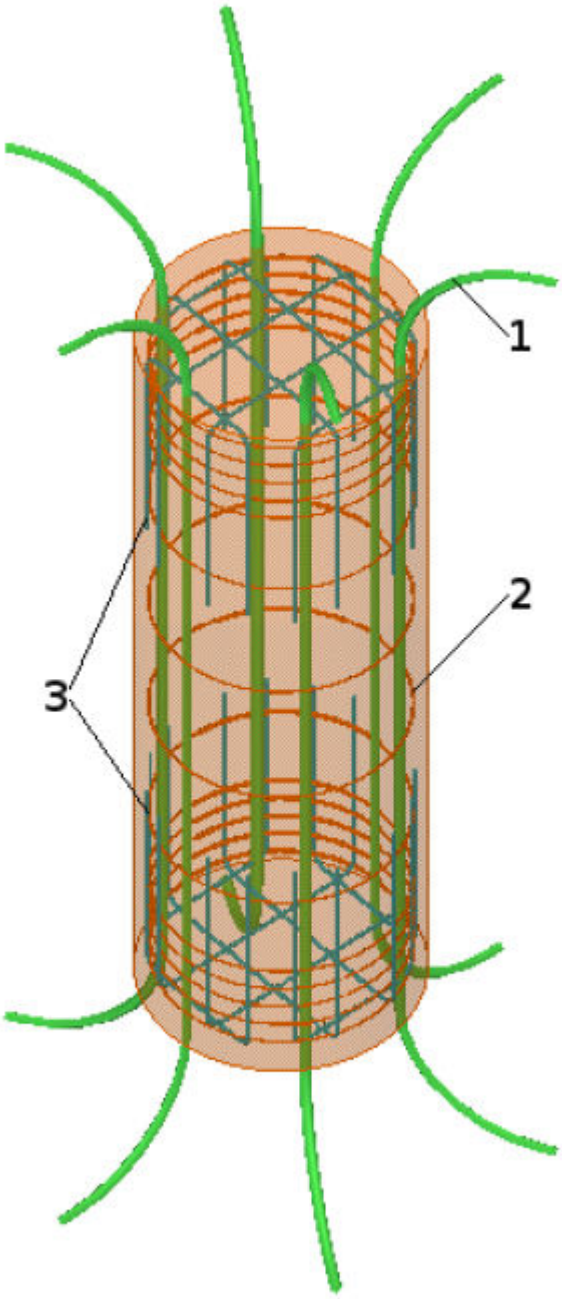
Do not use for rectangular columns.

Selection order

1. Select the main part (column).

   The component is created automatically when the part is selected.

Part identification key



	Part
1	Main bars
2	Stirrups
3	Column end reinforcement

Main bars tab

Use the **Main bars** tab to control the length of the vertical and horizontal extensions of the main bars at the top and bottom of the column, the thickness of the concrete cover over stirrups, and the cranking of the main bars.

Main bar properties

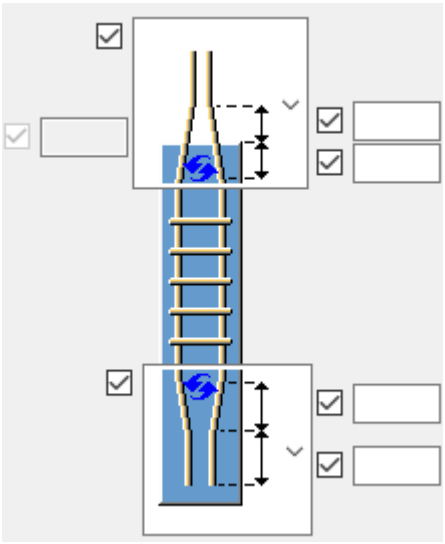
Option	Description
Number of bars	Number of main bars.
Rotation	Rotation of main bars.
Grade	Strength of the steel used in the reinforcing bars.
Size	Diameter of the main bars.
Bending radius	Bending radius of the main bars.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Name	Name for the main bars. Tekla Structures uses the name in drawings and reports.
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.

Hooks at top / bottom

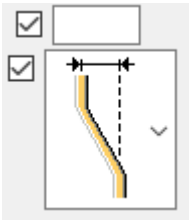
Option	Description
Type	Select the hook angle.
Length	Length of the hook.
Bending radius	Bending radius of the hook.

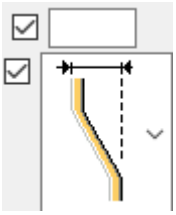
Cranking

You can create cranked reinforcing bars at the top and at the bottom of the column. Enter the vertical and sloped distance dimensions.



Enter the horizontal dimensions of top and bottom cranked bar as distance between center of bars or as outer distance of bars:

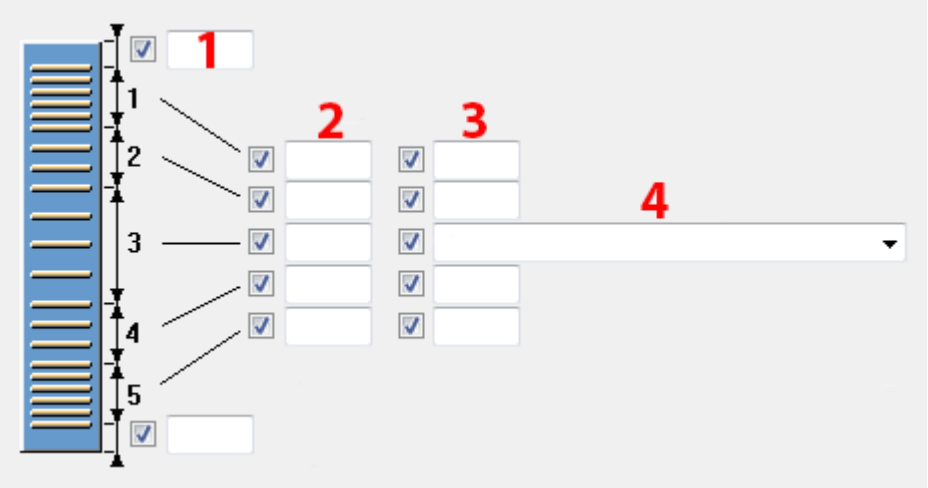




Stirrups tab

Use the **Stirrups** tab to control the thickness of the concrete cover over the stirrups at the top and bottom of the column, spacing and number of stirrups or laps in each stirrup group, stirrup types and stirrup shapes.

Stirrup dimensions



1. Thickness of the concrete cover over the stirrups at the top and the bottom of the column. The default cover thickness is 50 mm.

   Group 1 is the top stirrup group, 5 is the bottom stirrup group. Group 3 is always created.
2. Spacing of stirrups in each stirrup group.
3. Number of stirrups in each stirrup group.
4. Select how the stirrups are distributed.
   - **Exact space, flexible at ends**: Tekla Structures uses exactly the spacing value you specify, and evens out the stirrup distribution at the column ends.
   - **Target space**: Tekla Structures creates the stirrups at even spacings and tries to use the spacing value you specify.

Stirrup types

Option	Description
	Separate stirrups
	Single, continuous spiral stirrup
	Separate spiral stirrups

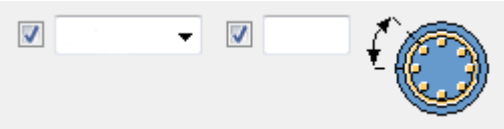
Option	Description
	Single, continuous stirrup

Stirrup shape

Select the shape of the stirrup from the list.

Stirrup overlap

Select the angle or the length of stirrup overlap. The option is not active if you have created spiral stirrups.



The overlapping angle can be of maximum 180 degrees.

The overlapping length can be in millimeters or inches.

Stirrup attributes tab

Use the **Stirrup attributes** tab to control the grade, size, class, name, and numbering properties of the stirrups.

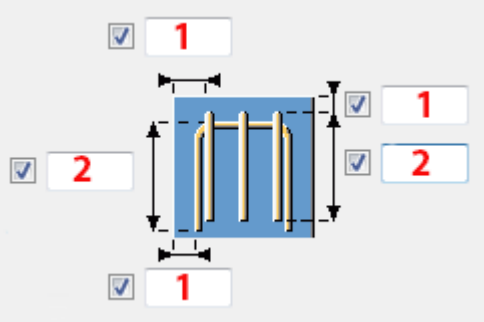
Stirrup properties

Option	Description
Grade	Strength of the steel used in the reinforcing bars.
Size	Diameter of the stirrups.

Option	Description
Name	Name for the stirrups. Tekla Structures uses the name in drawings and reports.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.

Top tab
Use the **Top** tab to control the thickness of the concrete cover of the top reinforcement, the number of top bars, and their spacing and rotation.

Column top reinforcement



1. Define the concrete cover thickness.
2. Define the length of the ultimate leg.

Column top reinforcement rotation

Select how to rotate the reinforcement at the top of columns.

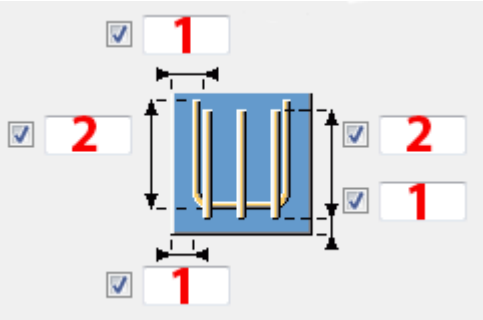
Column top reinforcement properties

Option	Description
Number of bars	Number of top bars.
Grade	Strength of the steel used in the reinforcing bars.
Size	Diameter of the top bars.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Option	Description
Name	Name for the top bars. Tekla Structures uses the name in drawings and reports.
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.

Bottom tab
Use the **Bottom** tab to control the thickness of the concrete cover of the bottom reinforcement, the number of bars, and their spacing and rotation.

Column bottom reinforcement



1. Define the concrete cover thickness.
2. Define the length of the ultimate leg.

Column bottom reinforcement rotation

Select how to rotate the reinforcement at the bottom of columns.

Column bottom reinforcement properties

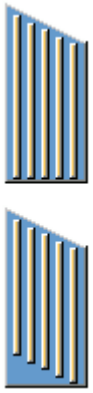
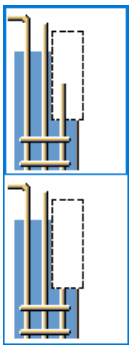
Option	Description
Number of bars	Number of bottom bars.
Grade	Strength of the steel used in the reinforcing bars.
Size	Diameter of the reinforcing bar.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Name	Name for the bottom bars. Tekla Structures uses the name in drawings and reports.
Prefix	Prefix for the part position number.

Option	Description
Start number	Start number for the part position number.

Advanced (main bars) tab

Use the **Advanced (main bars)** tab to control the main bar extensions separately for each bar.

Entensions

Option	Description
	In **Bar index**, enter the bar number for which to define the extension. Define the size, the distance from the column edge and the length of each extension. Select whether the main bars are tapered or moved according to the upper surface, if the top of the column is skewed.
	Select whether to cut bars by the cut in the column.

Rebar grouping

Select to create bars as a rebar group or as single bars from the **Creation method** list.

Rebar assembly tab

Use the **Rebar assembly** tab to add the created reinforcement as a rebar assembly to the cast units.

Create as rebar assembly

You can add the created reinforcement as a rebar assembly to the cast units with a predefined assembly type, name, prefix, and start number.

Option	Description
Create as rebar assembly	Select **Yes** to create all reinforcement as one rebar assembly, and to include it to the cast unit of the input part.
Rebar assembly type	Select the rebar assembly type. If you do not select the type, the default value of the rebar assembly is used.
Name, **Profile**, **Start number**	Define the name, profile, and start number. If you do not define these, the default values of the rebar assembly are used.

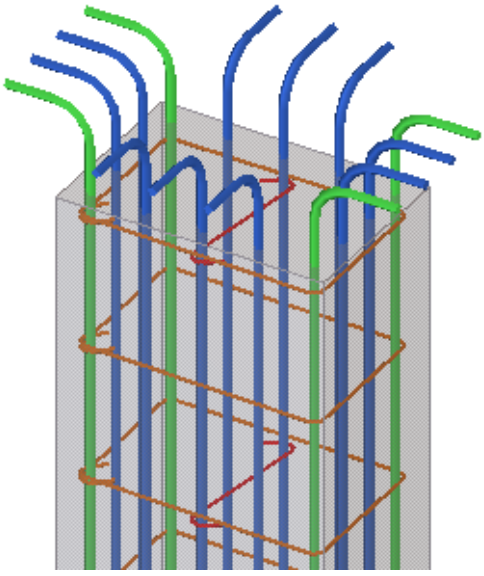
Rectangular column reinforcement (83)

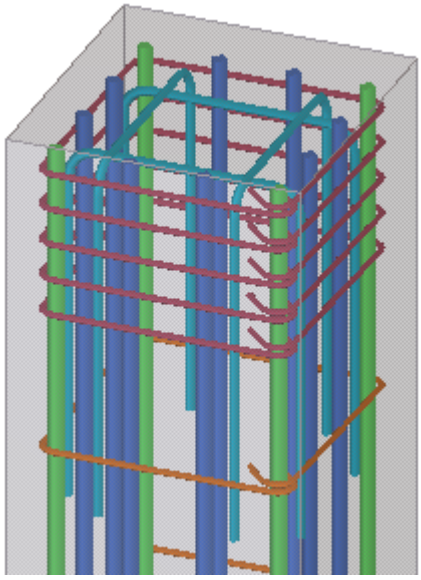
Rectangular column reinforcement (83) creates reinforcement for a concrete column that has a rectangular cross section.

Objects created

- Longitudinal main bars: corner bars (4), side bars
- Stirrups
- Intermediate links
- Column end reinforcement

Use for

Situation	Description
	Rectangular concrete column with corner bars and side bars bent outside the column. Side bars on long sides. Intermediate links tie side bars at every second stirrup.

Situation	Description
	Rectangular concrete column with straight corner and side bars entirely inside the column. End of column reinforced.

Limitations

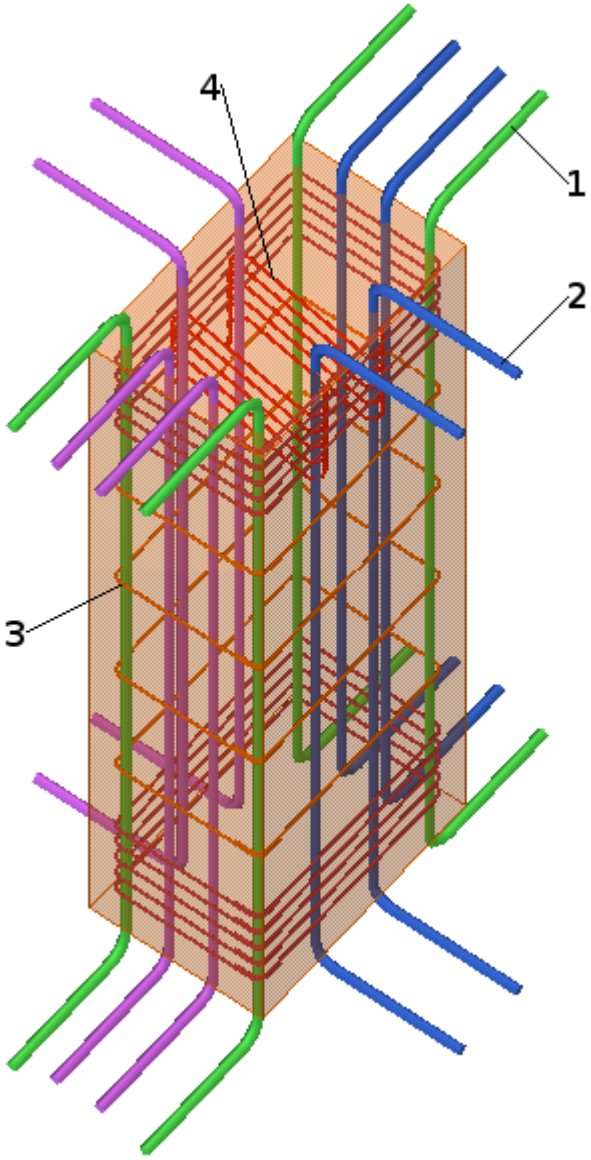
Do not use for round columns.

Selection order

1. Select the main part (column).

   The component is created automatically when the part is selected.

Part identification key



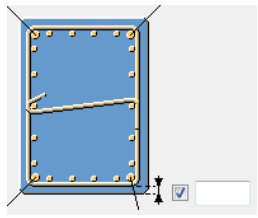
	Part
1	Corner bars
2	Side bars
3	Stirrups
4	Intermediate links

Main bars tab

Use the **Main bars** tab to control the corner bar properties, the symmetry options, rotation, and concrete cover thickness.

Basic corner bar properties

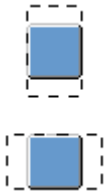
Define the grade, size and bending radius of the corner bars. The active settings depend on the selected symmetry option.

Option	Description
Grade	Strength of the steel used in the reinforcing bars.
Size	Diameter of the reinforcing bar.
Bending radius	Define the bending radius of the corner bars.
	Define the concrete cover thickness. Select whether the thickness is the same on all sides.

Symmetry options

Select the symmetry option. Use the symmetrical conditions to define which of the corner bars have the same grade, size, and bending radius properties. The corner bars that have same properties are symmetrical.

Rotation



In square columns, you can select the perpendicular sides of a column if the sides require different reinforcement. You can rotate all reinforcement in a square column by 90 degrees.

Additional corner bar properties

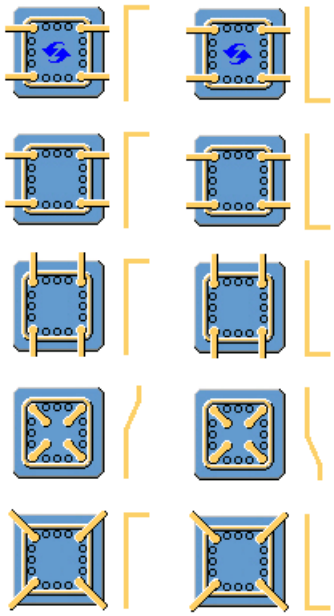
Option	Description
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Name	Define a name for the main bars. Tekla Structures uses the name in drawings and reports.

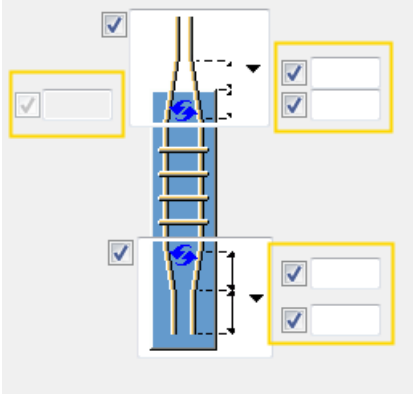
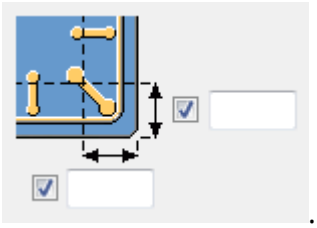
Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.

Bar ends tab

Use the **Bar ends** tab to control the length of vertical and horizontal extensions for the corner bars and side bars, and the cranking.

Bar creation	Select whether the extensions of corner bars and side bars are created symmetrically on both sides of the column. If you select **Not symmetrical**, you can enter separate extension values for the opposite sides of the column.
Vertical extension	Define the length of the vertical extension outside the column for corner bars and side bars. The active settings depend on **Bar creation**: **Symmetrical** or **Not symmetrical**.
Horizontal extension	Define the length of the horizontal extension for corner bars and side bars. The active settings depend on **Bar creation**: **Symmetrical** or **Not symmetrical**.
Corner bars **Side bars 1** **Side bars 2**	Select the hook type, and define the hook length and bending radius.

Top corner bar direction **Bottom corner bar direction**	Select the direction of the corner bars.  The options illustrate whether the direction is a horizontal extension or a cranking option.
Cranking	You can create cranked reinforcing bars at the top and at the bottom of the column. To activate the cranking options, select in the **Top corner bar direction** and **Bottom corner bar direction** lists. Define the dimensions for the cranked bars.

	 To successfully create the cranked reinforcing bars, ensure that the bending radius is not too large.
Align cranking	When set to **Yes**, the cranking of the corner bars is automatically aligned to the horizontal extension direction. This helps you to output two-dimensional bars easier.
Edge dimensions	Define the edge distance from the column corner to the point where the cranking starts  Select cranking from the edge or center line of the rebar: Define the cranking separately for the corner bars and the side bars.

Side bars tab
Use the **Side bars** tab to control the number of side bars, side bar spacing and placing, symmetry options, and properties.

Number of side bars

Define the number and spacing of side bars. You can define two sets of side bars on each side of the column.

You can define the side bars separately for each side of the column.

The active settings depend on the selected symmetry option.

Placing of side bars

Select the horizontal and vertical placing for the side bars. Select whether the bars are placed starting from the corner or at equal spaces.

	Distance between corner bars and side bars.
	Distance between side bars.

Symmetry options

Select the symmetry option. Using the symmetrical conditions you can define which side bars are symmetrical, and which side bars use the same properties.

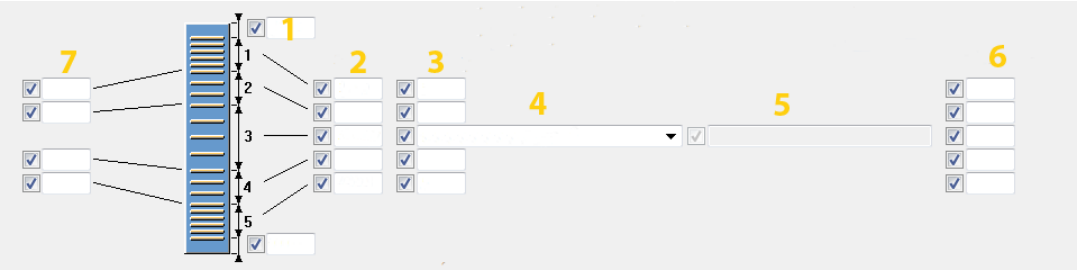
Side bar properties

Option	Description
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Bending radius	Define the bending radius of the side bars.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Option	Description
Name	Define a name for the side bars. Tekla Structures uses the name in drawings and reports.
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.

Stirrups tab
Use the **Stirrups** tab to control the stirrup properties.

Stirrup dimensions



1	Define the thickness of the concrete cover over the stirrups at the top and the bottom of the column. The default cover thickness is `50` mm. Group **1** is the top stirrup group, **5** is the bottom stirrup group. Group **3** is always created.
2	Define the spacing of stirrups in each stirrup group.
3	Define the number of stirrups in each stirrup group.
4	Select how the stirrups are distributed.
5	If you select the **Distance list** option, enter different spacing values for the groups.
6	Define the cover thickness for each stirrup group.
7	Define the gaps between the stirrup groups.

Create stirrups

Select whether the stirrups are created as individual reinforcing bars, rebar group, or spiral rebar group.

Rebar lapping at stirrup corners

Select the how the rebars lap at the stirrup corners. The options are 135-degree hooks or 90-degree hooks at the bar end, or overlapping U-shape stirrups.

You can define the overlap length for the U-shape stirrups.

Ignore cuts

If you have a recess or a hole in the column, you can select to ignore the cuts at the top and the bottom of the column when the stirrups are created.

Stirrup properties

Option	Description
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Name	Define a name for the stirrups. Tekla Structures uses the name in drawings and reports.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.
Overlap length	Define the overlap length for the U-shape stirrups.

Intermediate links tab

Use the **Intermediate links** tab to control the intermediate links to tie all side bars.

If you have rectangular columns that have very large cross sections, the side bars may be far from the corners of the stirrups. You will need to create

intermediate links to tie all side bars, and to prevent them from buckling when they are in compression.

Intermediate links are created for each stirrup group.

NOTE Intermediate links are created between **Side bars 1** or **Side bars 2** that are symmetrical.

For **Side bars 2** intermediate links are created only if no **Side bars 1** are created.

Create as one group

Select whether all the stirrups are created as a single group.

Properties of intermediate links

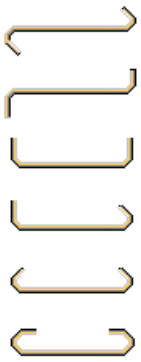
Option	Description
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Name	Define a name for the stirrups. Tekla Structures uses the name in drawings and reports.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.

Spacing of intermediate links

	Spacing is the same as stirrup spacing.
	Spacing is double the stirrup spacing (intermediate links at every second stirrup).
	Same as above, but for alternate stirrups.
	No intermediate links are created.

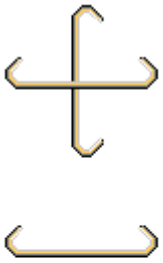
Intermediate link type

Select the type of the link.



Intermediate link pattern

Select whether intermediate links go in one direction or in a crossing pattern.

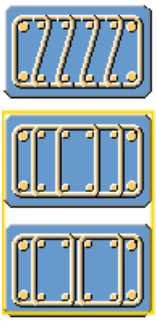


Hook orientation

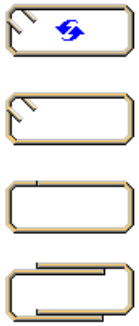
Select the hook orientation for both sides.

Closed stirrups as intermediate links

Select whether to use closed stirrups as intermediate links.



If you select closed stirrups, select the type of the bar lapping at the stirrup corners.



Zone without intermediate links

Define the zone length where intermediate links are not created. In this zone, the stirrups tie the side bars. The distance is measured from the stirrup corner.

If you have selected to use closed stirrups, you can define the distance from the corner of the closed stirrups.

Top / Bottom tabs

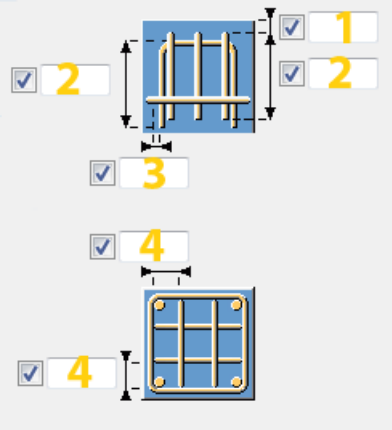
Use the **Top** and **Bottom** tabs to control the top and bottom reinforcement thickness of the concrete cover, number of bars, spacing and rotation.

Number and spacing of bars

Define the number and spacing of the top or bottom bars.

You can define the number of transverse bars in both cross directions.

Column top / bottom reinforcement



	Description
1	Define the concrete cover thickness from the top / bottom of the column.

	Description
2	Define the length of the rebar legs.
3	Define the concrete cover thickness from the sides of the column.
4	Define the distance from the edge of the column to the edge of the first rebar in the group.

Placing of top or bottom bars

Select the horizontal and vertical placing for the top or bottom bars. Select whether the bars are placed starting from the corner or at equal spaces.

	Distance between corner bars and side bars.
	Distance between side bars.

Rotation

Select how to rotate the reinforcement at the top or bottom of columns.

	No rotation. Transverse bars are perpendicular to the longer side of the column.
	Rotation angle is 90 degrees. Transverse bars are parallel to the longer side of the column.

Top and bottom reinforcement properties

Option	Description
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Bending radius	Define the bending radius of the top or bottom bars.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Name	Define a name for the top or bottom bars. Tekla Structures uses the name in drawings and reports.
Prefix	Define a prefix for the part position number.

Option	Description
Start number	Define a start number for the part position number.

Rebar assembly tab

Use the **Rebar assembly** tab to add the created reinforcement as a rebar assembly to the cast units.

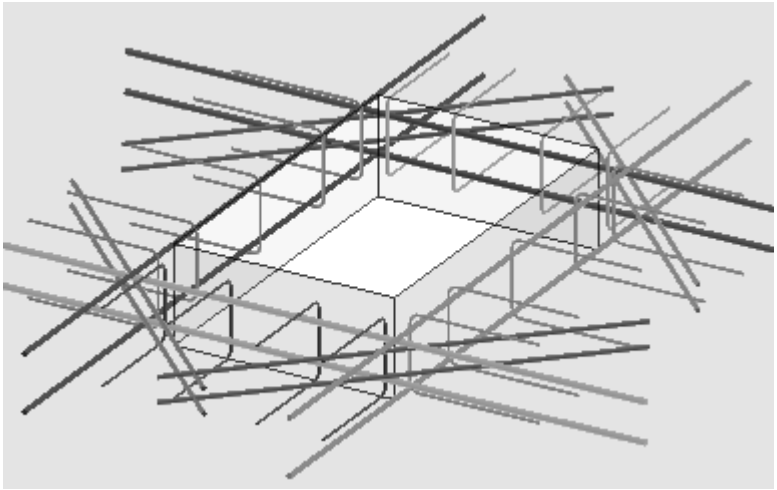
Create as rebar assembly

You can add the created reinforcement as a rebar assembly to the cast units with a predefined assembly type, name, prefix, and start number.

Option	Description
Create as rebar assembly	Select **Yes** to create all reinforcement as one rebar assembly, and to include it to the cast unit of the input part.
Rebar assembly type	Select the rebar assembly type. If you do not select the type, the default value of the rebar assembly is used.
Name, **Profile**, **Start number**	Define the name, profile, and start number. If you do not define these, the default values of the rebar assembly are used.

Hole reinforcement for slabs and walls (84)

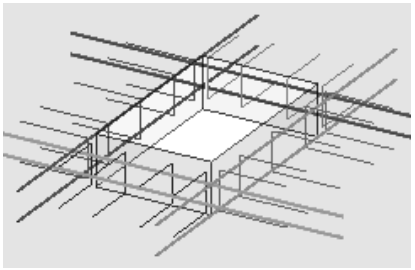
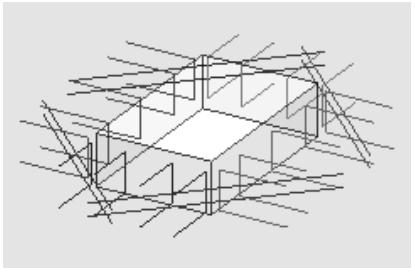
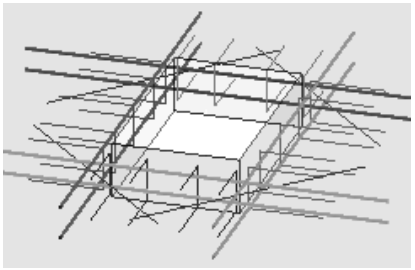
Hole reinforcement for slabs and walls (84) creates a hole in a concrete slab or wall and reinforcement around the hole.

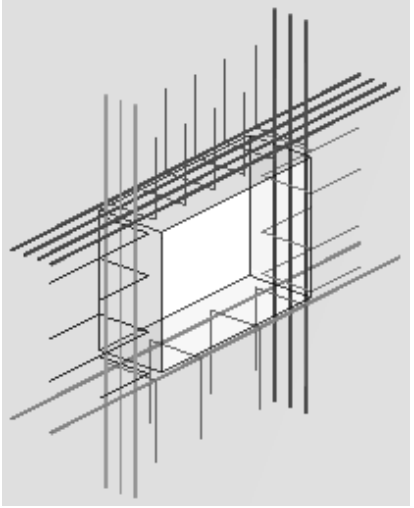


Bars created

- Straight bars along hole edges
- Diagonal bars close to hole corners
- U-shaped edge bars

Use for

Situation	More information
Rectangular or round holes in concrete slabs and walls	
	Straight and edge bars only, no diagonal bars.
	Diagonal and edge bars only, no straight bars.
	Hole rotated from the direction of the slab. One diagonal bar at each corner.

Situation	More information
	Different number of bars on each side of the hole in the wall. No diagonal bars.

Before you start

- Create the concrete slab or wall.
- Calculate the required area of reinforcement.

Selection order

1. Select the center of the hole.
2. Select the concrete slab or wall.

Picture tab

Use the **Picture** tab to define creation of the hole, hole and bar dimensions, and concrete cover thickness.

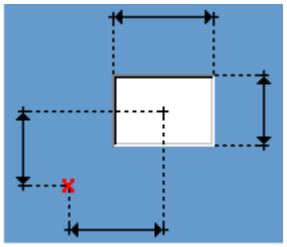
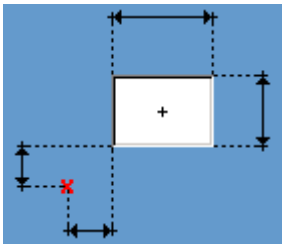
Hole

Select to create a hole and reinforce it or to create reinforcements to an existing hole from the **Create** list.

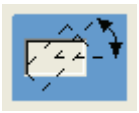
Select the shape of the hole:

Option	Description
	Rectangular
	Round

Select the pivot point for hole offsets:

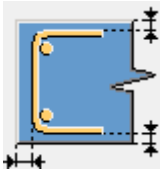
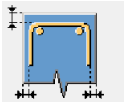
Option	Description
	Hole center
	Hole corner

Enter the rotation angle of the hole:



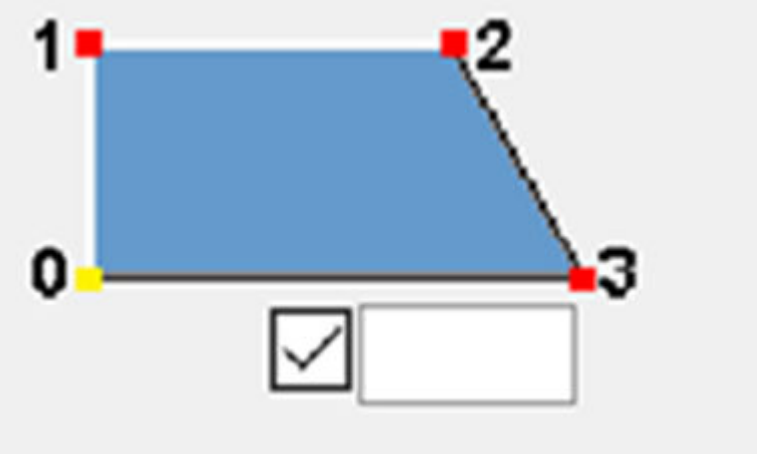
Concrete cover

Enter the cover thicknesses for horizontal and vertical edge bars.

Horizontal	Vertical
	

Generation start point

You can specify which point of the slab is taken as the origin where the creation plane starts. This means that the created reinforcement is aligned to the slab edge starting from that point.



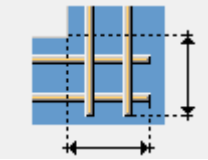
Horizontal and vertical bars tab
Use the **Horizontal and vertical bars** tab to define which bars are closest to the surface of the concrete, and the properties of bar groups along the sides of the hole.

The slab or wall direction defines the direction of the hole and which bars are to the left and right of, and above and below the hole.

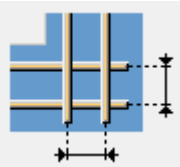
Option	Description
Grade	Strength of the steel used in the reinforcing bars.
Size	Diameter of the reinforcing bar.
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Define the name for the reinforcing bars. Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Additional bars tab
Use the **Additional bars** tab to create additional bars and define their properties.

Enter the length of additional bars from the opening corner.



Enter the additional bars offsets from horizontal and vertical bars.



Option	Description
Grade	Strength of the steel used in the reinforcing bars.
Size	Diameter of the reinforcing bar.
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Edge bars tab

Use the **Edge bars** tab to define the properties of bar groups on each side.

The slab or wall direction defines the direction of the hole and which bars are to the left and right of, and above and below the hole.

Option	Description
Grade	Strength of the steel used in the reinforcing bar.
Size	Diameter of the reinforcing bar.
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.

Option	Description
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Diagonal bars tab
Use the **Diagonal bars** tab to define the properties of bar groups at each corner of the hole.

The slab or wall direction defines the direction of the hole and which bars are to the left and right of, and above and below the hole.

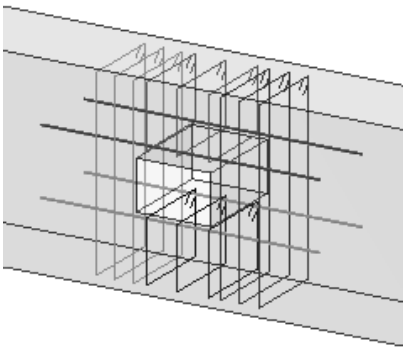
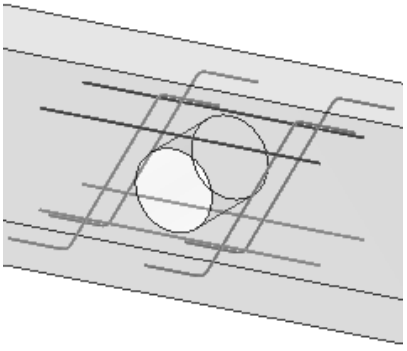
Option	Description
Grade	Strength of the steel used in the reinforcing bars.
Size	Diameter of the reinforcing bar.
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Hole creation and reinforcement (85)
Hole creation and reinforcement (85) creates a hole in a concrete part and reinforcement around the hole.

Bars created

- Straight bars at hole edges
- Stirrups
- Z-shaped bars

Use for

Situation	More information
	Rectangular hole, stirrups on each side of the hole, no Z-shaped bars.
	Round hole, Z-shaped bars, no stirrups around the hole.
Concrete beams or columns	
Round or rectangular holes	Hole only. No additional reinforcement around it.
With or without Z-shaped bars or stirrups	

Do not use for

Parts that have round or irregular cross sections.

Before you start

- Create the concrete part.
- Calculate the required area of reinforcement.

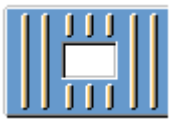
Selection order

1. Select the center of the hole.
2. Select the concrete part.

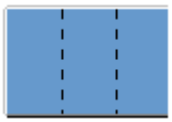
Picture tab
Use the **Picture** tab to define hole properties, bar dimensions and locations, concrete cover thickness, and angle of Z-shaped bars.

Holes

Use these options to create holes and additional reinforcement:

Option	Description
	Creates a hole and additional reinforcement around it.
	Only creates a hole, no reinforcement.

Use these options to define the direction of holes:

Option	Description
	Hole along the local y direction of the part.
	Hole along the local z direction of the part.

Z-shape bars

Use these options to define Z-shaped bars around round holes:

Option	Description
	Concrete cover measured in the direction of the radius of the hole.
	Concrete cover measured from the corner of the bounding box around the hole.

Use these options to define the direction of Z-shaped bars:

Option	Description

Bars tab

Use the **Bars** tab to define bar properties.

Option	Description
Grade	Strength of the steel used in the reinforcing bars.
Size	Diameter of the reinforcing bar.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.

Stirrups tab

Use the **Stirrups** tab to define stirrups properties and additional settings.

Stirrups

Use these options to define the stirrups around holes:

Option	Description
	Single stirrups
	Bundled stirrups

Hooks

Option	Description
	135-degree hooks
	90-degree hooks

Braced girder (88)

Braced girder (88) creates braced girders in a precast concrete part, such as in a thin-shell slab or in a sandwich wall panel. The girders are cast into the concrete part and they also work as a part of the reinforcement and as a connector between the precast and cast-in-place concrete.

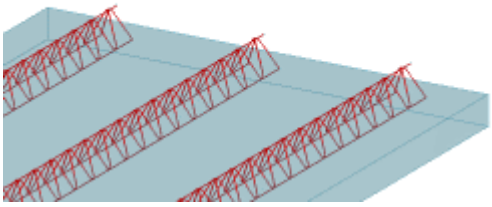
Objects created

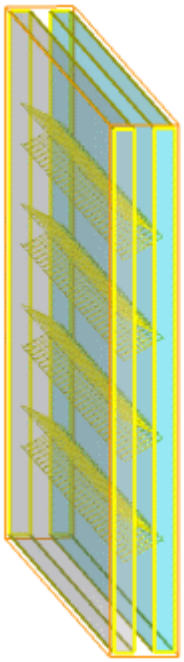
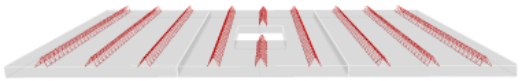
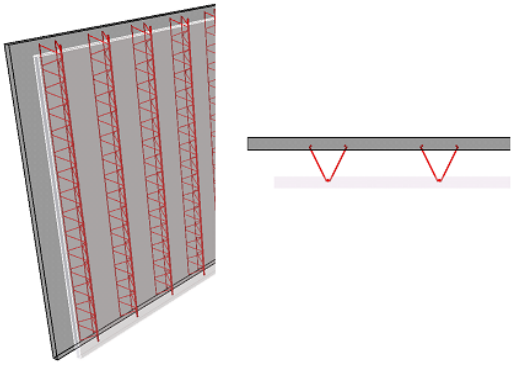
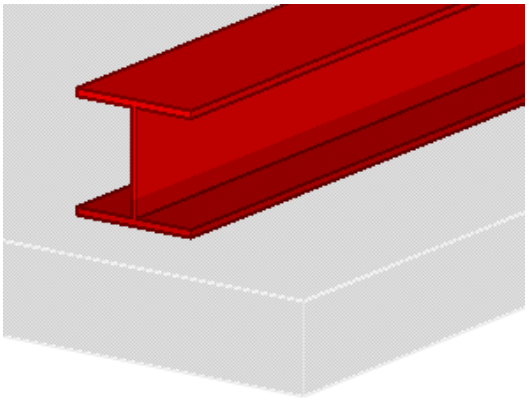
The girders consist of the following parts:

- Two bottom reinforcing bars
- One or two top reinforcing bars
- Two connecting reinforcing bars

Instead of using reinforcing bars, you can use profiles and plates to create the braced girders.

Use for

Situation	Description
	Braced girders are created in the precast concrete slab.

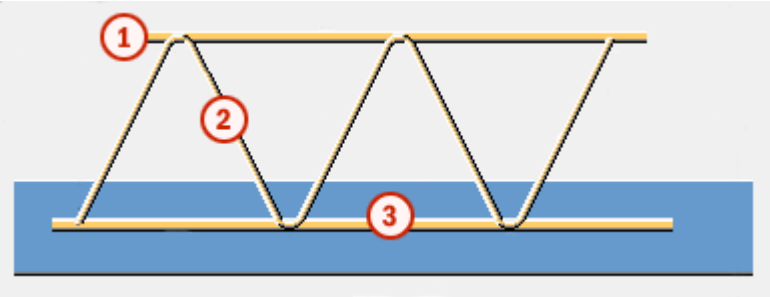
Situation	Description
	Braced girders are created in the precast concrete sandwich panel.
	Braced girders are created in a wide plate floor with an opening.
	Braced girders are created in a hollow wall.
	Profile as a girder.

Selection order

1. Select the concrete part.

   The girders are created automatically.

Part identification key

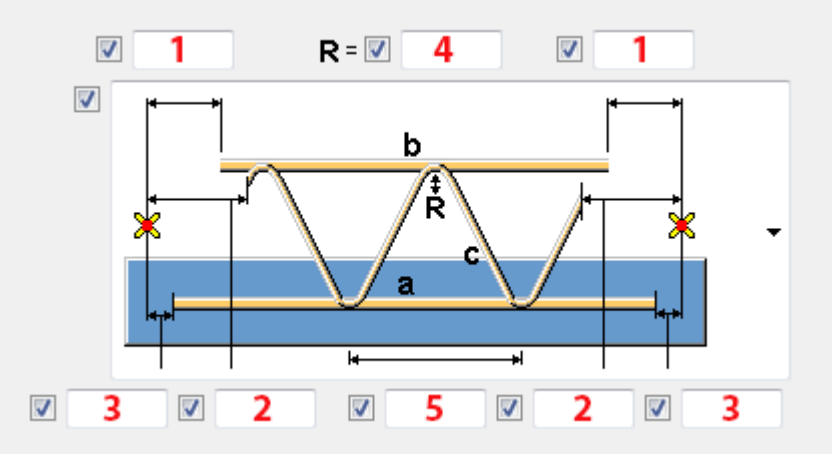


	Part
1	Top reinforcing bar
2	Connecting reinforcing bar
3	Bottom reinforcing bar

Picture tab

Use the **Picture** tab to control the creation, shape and dimensions of reinforcing bars.

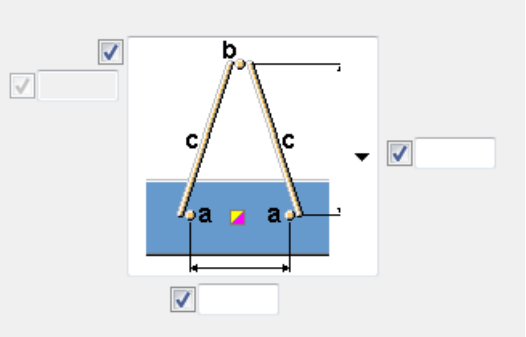
Reinforcing bar dimensions and shape



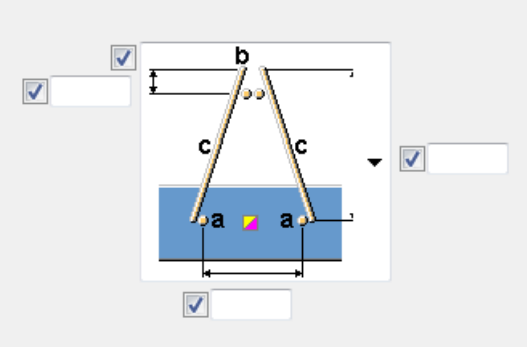
	Description
1	Distance between the end of the top reinforcing bar and the part end.
2	Distance between the end of the connecting reinforcing bar and the part end.

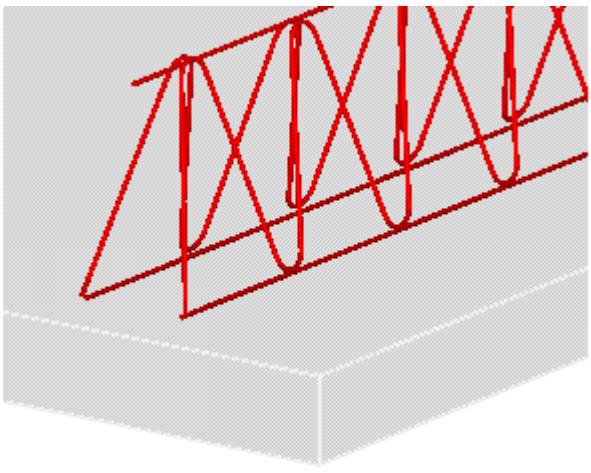
	Description
3	Distance between the end of the bottom reinforcing bar and the part end.
4	Radius of the connecting reinforcing bar.
5	Distance between bendings.

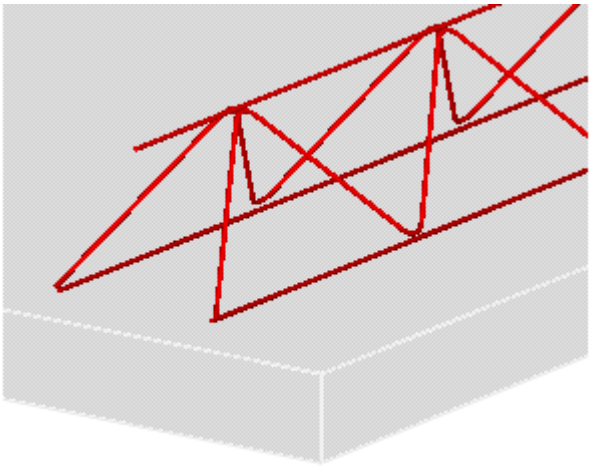
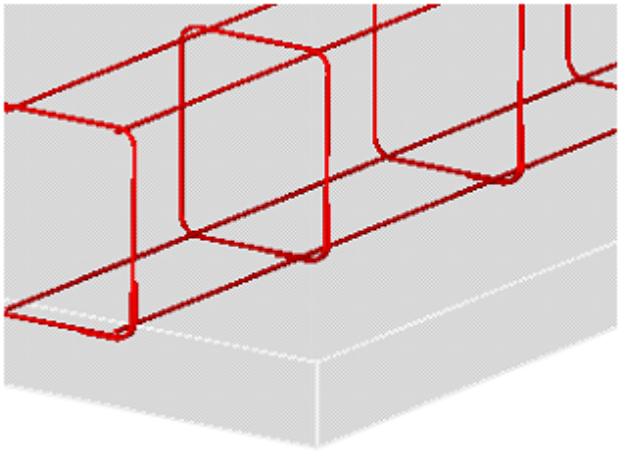
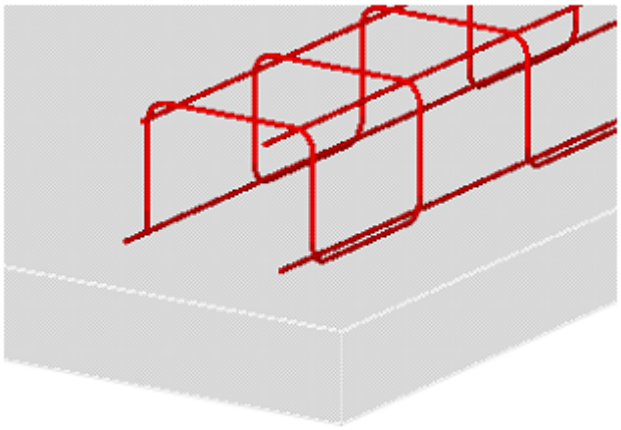
Define the distance between the bottom reinforcing bars, and the distance between the top and bottom reinforcing bars.



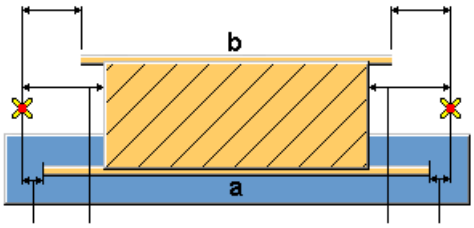
If you add two reinforcing bars at the top, you can define the distance of these reinforcing bars from the top of the connecting reinforcing bars.



Examples


Examples




Girder creation

Option	Description
Create	Select the type of the bottom, top and connecting reinforcing bars (Bar a, b, c): • **Reinforcing bar** • **Steel rod** • **No** (The reinforcing bar is not created.)
Profile	Profile selection is activated when you select the following girder option: Define a prefix and a start number for the part position number, and material, name, comment, and class.
Plate	Plate creation is activated when you select the following girder option:  Define a prefix and a start number for the part position number, and material, name, comment, and class.
Add as	Select the method that is used to connect the girders to the cast unit: **Sub-assembly**, **Welded**, **No**

Reinforcing bar properties

Define the reinforcing bar properties for the top, bottom and the connecting reinforcing bars.

Option	Description
Size	Size of the reinforcing bar.
Grade/Material	Grade of the reinforcing bar.

Option	Description
Name **Prefix** **Start number** **Comment** **Class**	A name, a prefix and a start number for the part position number, and comment and class for the reinforcing bar.

Create as rebar assembly

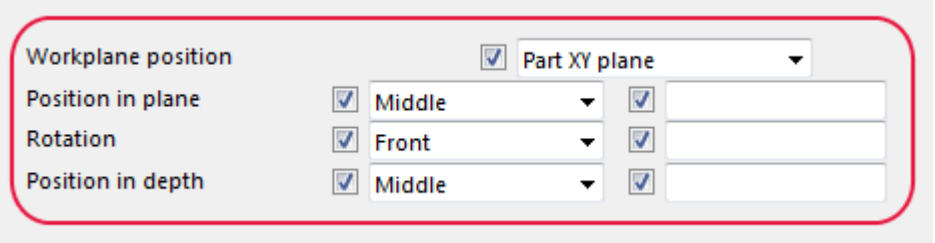
You can add the created reinforcement as a rebar assembly to the cast units with a predefined assembly type, name, prefix, and start number.

Option	Description
Create as rebar assembly	Select **Yes** to create all reinforcement as one rebar assembly, and to include it to the cast unit of the input part.
Add to existing rebar assembly	• **Do not add**: New rebar is added as a rebar assembly to the cast unit. • **As single bars**: Add all new rebars directly into an existing rebar assembly as rebars. • **As sub-assembly**: Add all new rebars to their own new rebar assembly, which is then included as a sub-assembly to the main rebar assembly.
Rebar assembly type	Select the rebar assembly type. If you do not select the type, the default value of the rebar assembly is used.
Name, **Profile**, **Start number**	Define the name, profile, and start number. If you do not define these, the default values of the rebar assembly are used.

Parts tab

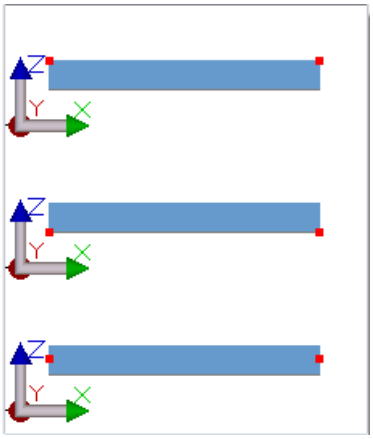
Use the **Parts** tab to control the positioning and number of girders.

Workplane orientation



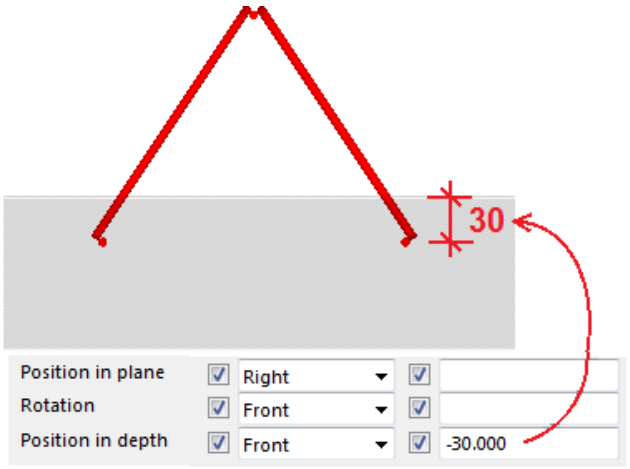
Option	Description
Workplane position	Use this setting to control to which side of the input part the girders are created. The **Model** option creates them according to the current work plane in the model. • **Part XY plane** • **Part YZ plane** • **Part ZX plane** • **Model**
	Use the **Position in plane**, **Rotation** and **Position in depth** options to define the orientation of the girder on the work plane.

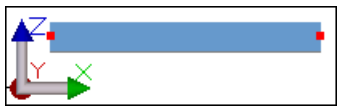
Girder position in z direction

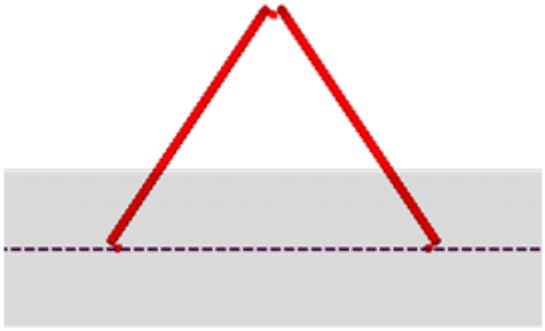


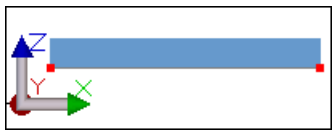
Select how the girders are positioned in the z direction.

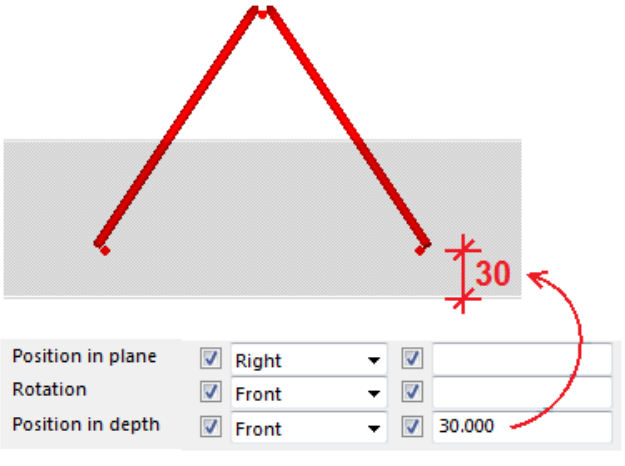
Examples



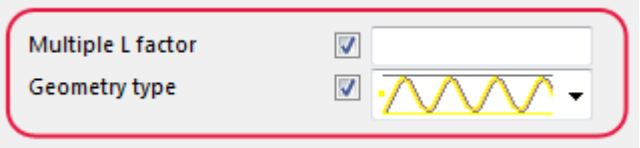


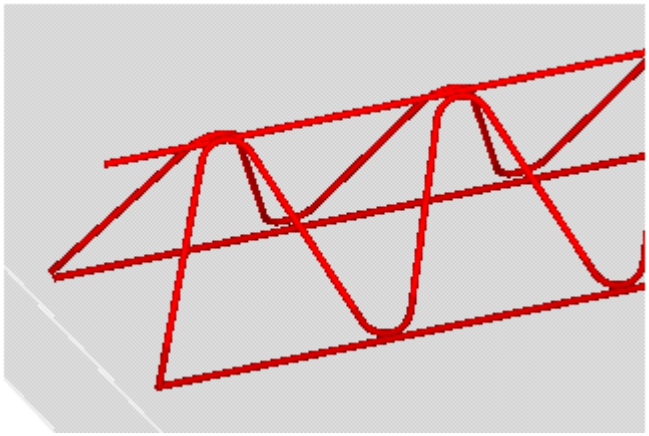
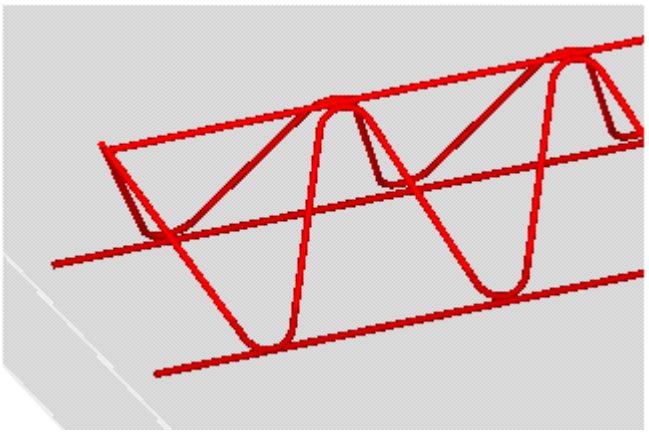


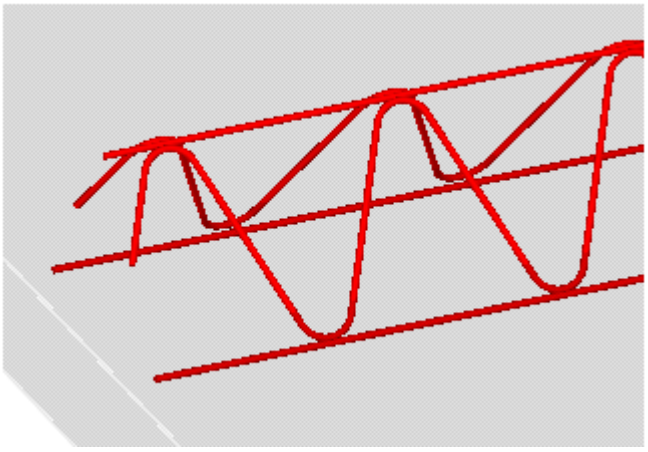




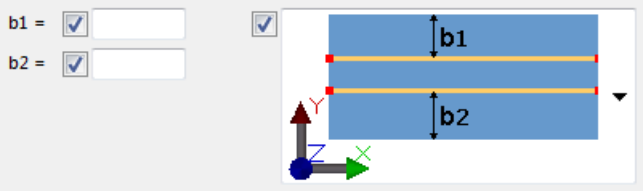
Geometry



Option	Description
Multiple L factor	Define the accuracy for the rounding of the girder length. The default value is 1.0. With the default value, there are no decimals in the girder length.
Geometry	Select the geometry for the connecting reinforcing bars. Examples:  

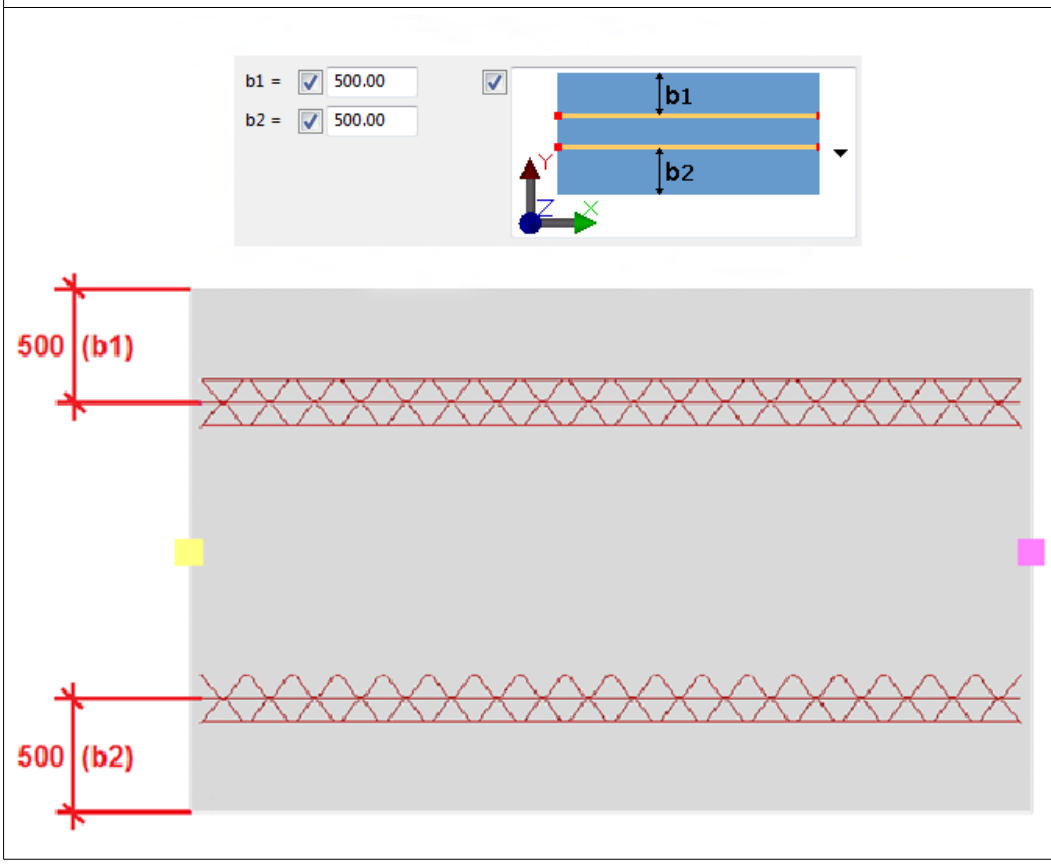
Option	Description
	

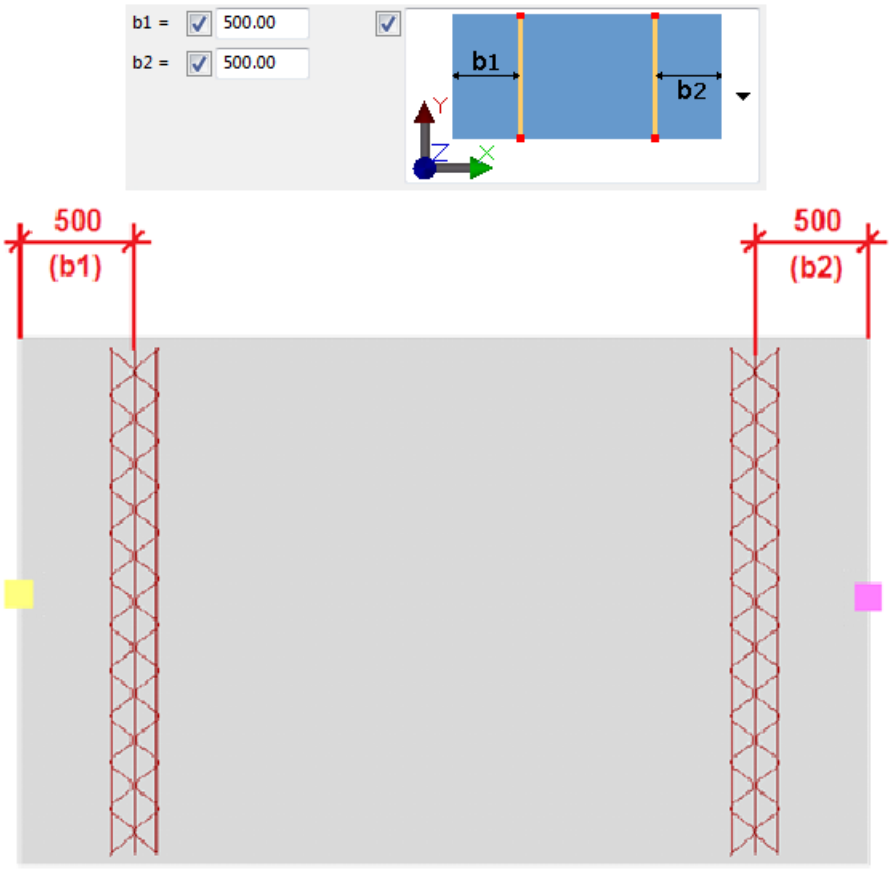
Girder position in y direction



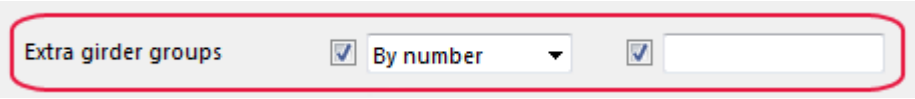
Select how the girders are positioned in the y direction.

Examples

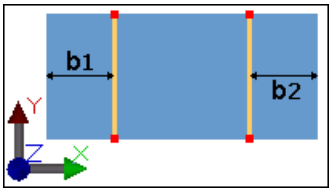


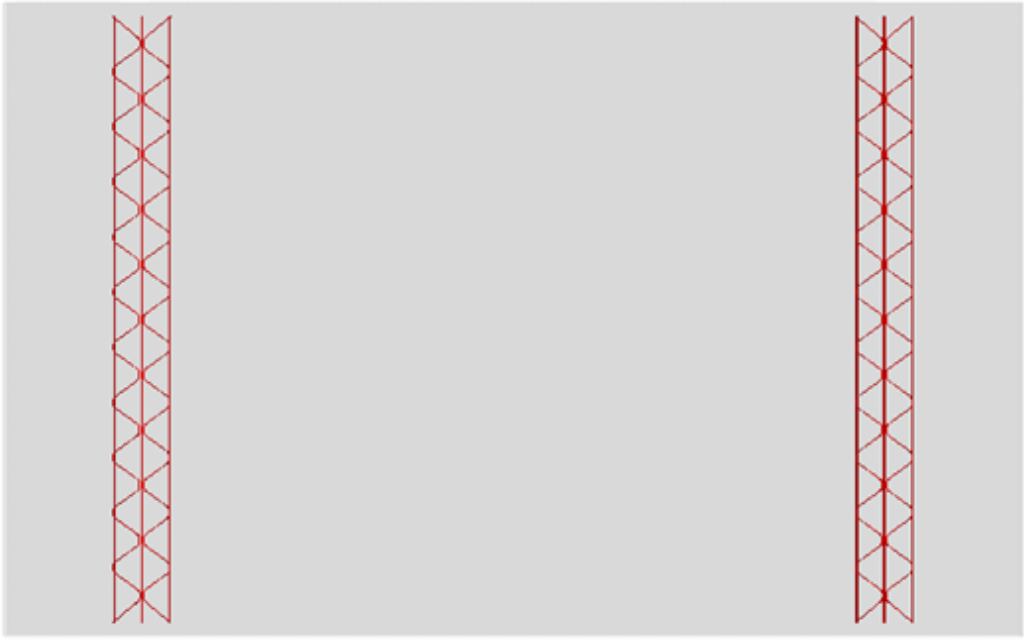
Examples


Extra girders



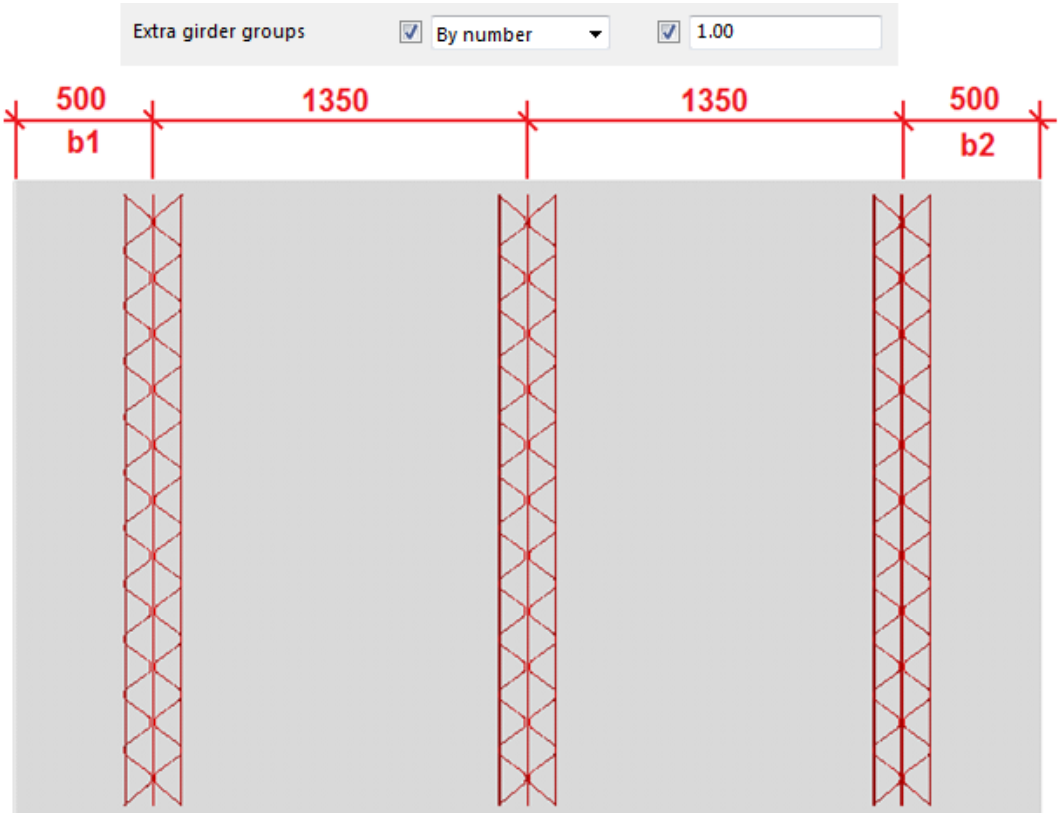
Select whether additional girder groups are created.

Examples
For example, if you have selected this option: 

Examples
Extra girder groups is set to **None**: No additional girder groups are created between 2 existing braced girders. 

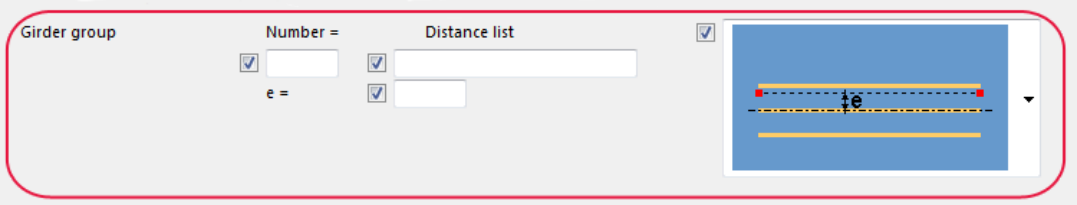
Examples

Extra girder groups is set to **By number**: Additional girder groups are created based on the entered number. Distances between the groups are equally divided.



Examples
Extra girder groups is set to **By distance**: The number of additional girder groups is based on the defined distance. The distances between the girder groups are equally divided.

Girder group

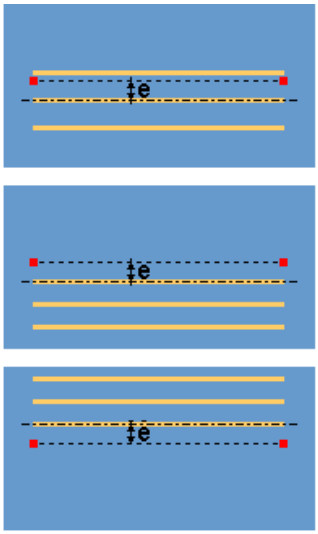


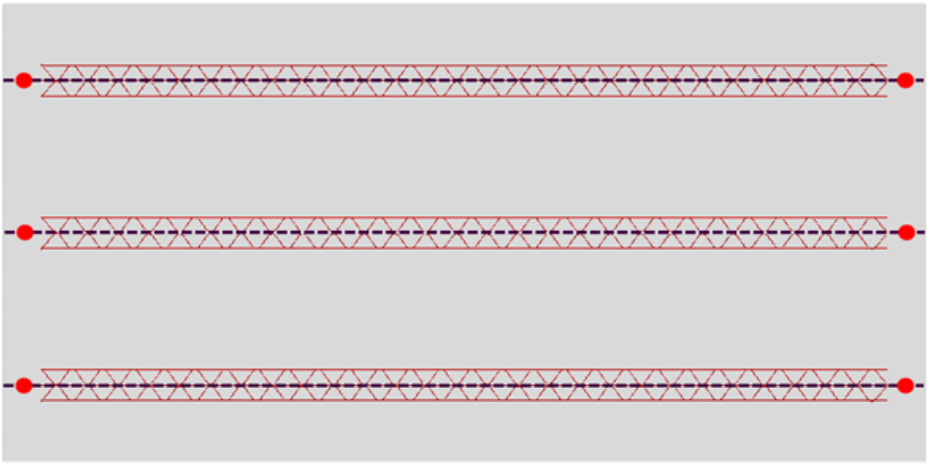
Define whether additional girders are created from the existing girders.

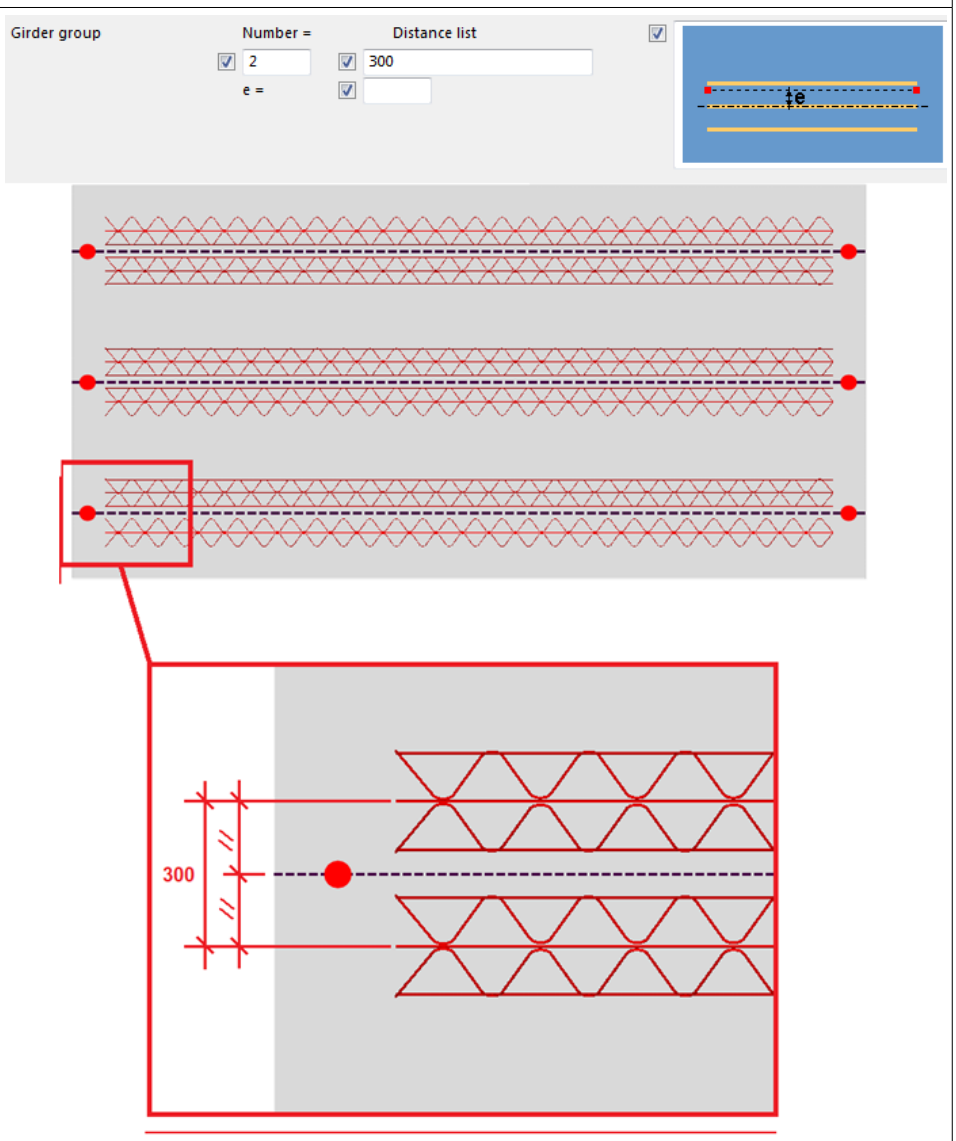
Number is the number of girders in the girder group.

Distance list is the distance between the girders in the girder group.

e = is the position of the girders from the reference line.

Option	Description
	• Girder group is positioned in the middle of the reference line. • Girder group is positioned on the right side of the reference line. • Girder group is positioned on the left side of the reference line.

Examples	
Existing girders	

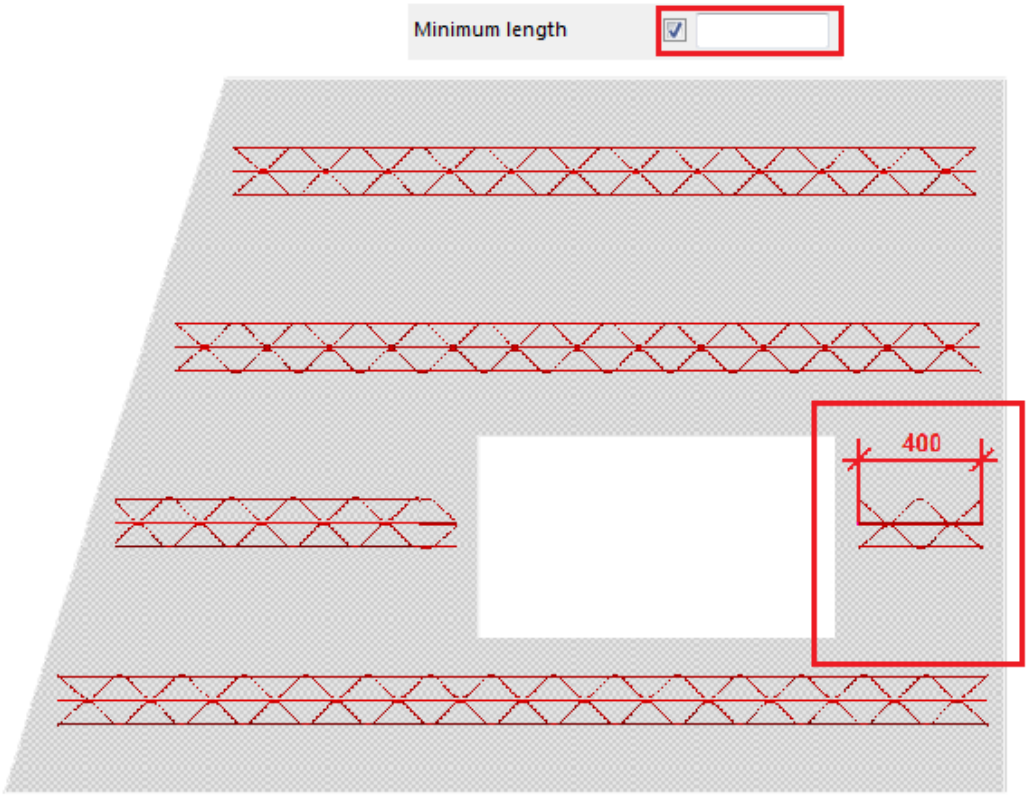
Examples	
Additional girders	

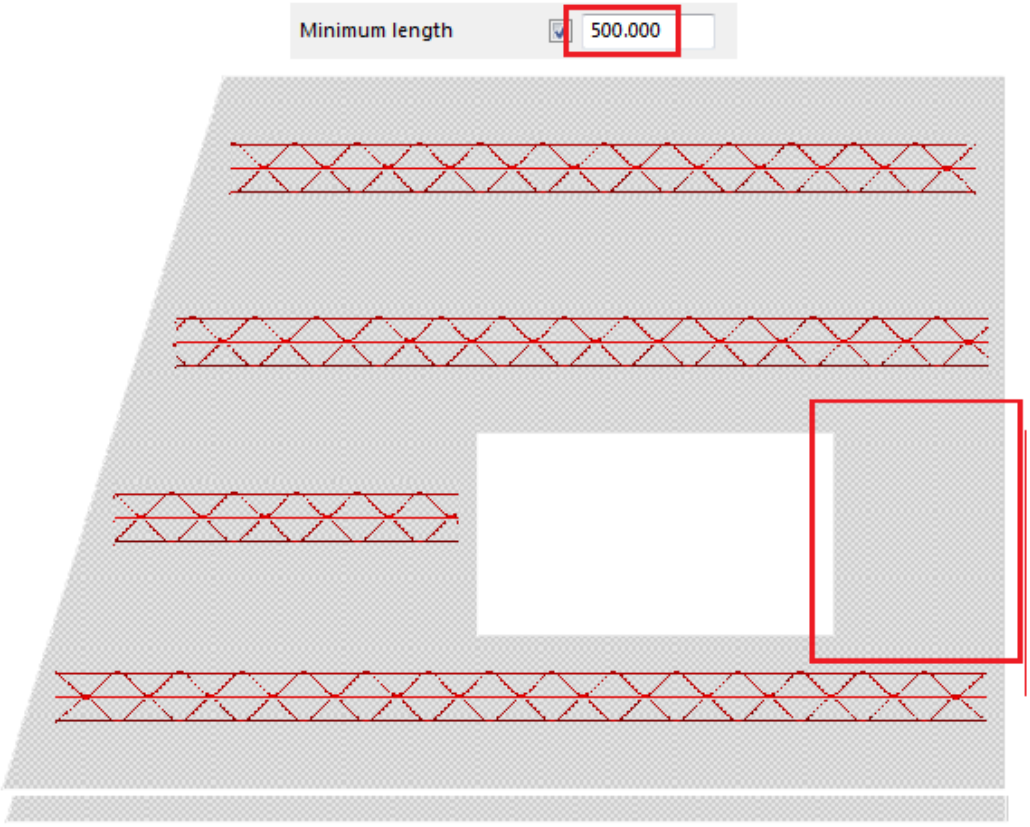
Geometry tab

Use the **Geometry** tab to control how openings and part length affect the girder creation.

Option	Description
Create always girders	Select whether girders are always created. If you select **Yes**, girders are created even when the girder is placed fully outside the concrete part.
Openings	Select whether girders are created in openings.

Option	Description
Length	Select how girders adapt to the part shape.
Minimum length	Define the minimum length of the girder.
Maximum length	Define the maximum length of the girder. The girder is split when the maximum length is reached.

Examples of minimum length
No minimum length defined: 

Examples of minimum length
Defined minimum length (no girder created): 

Double wall tab
Use the **Double wall** tab to select how a second concrete element affects the girders in **Braced girder (88)**.

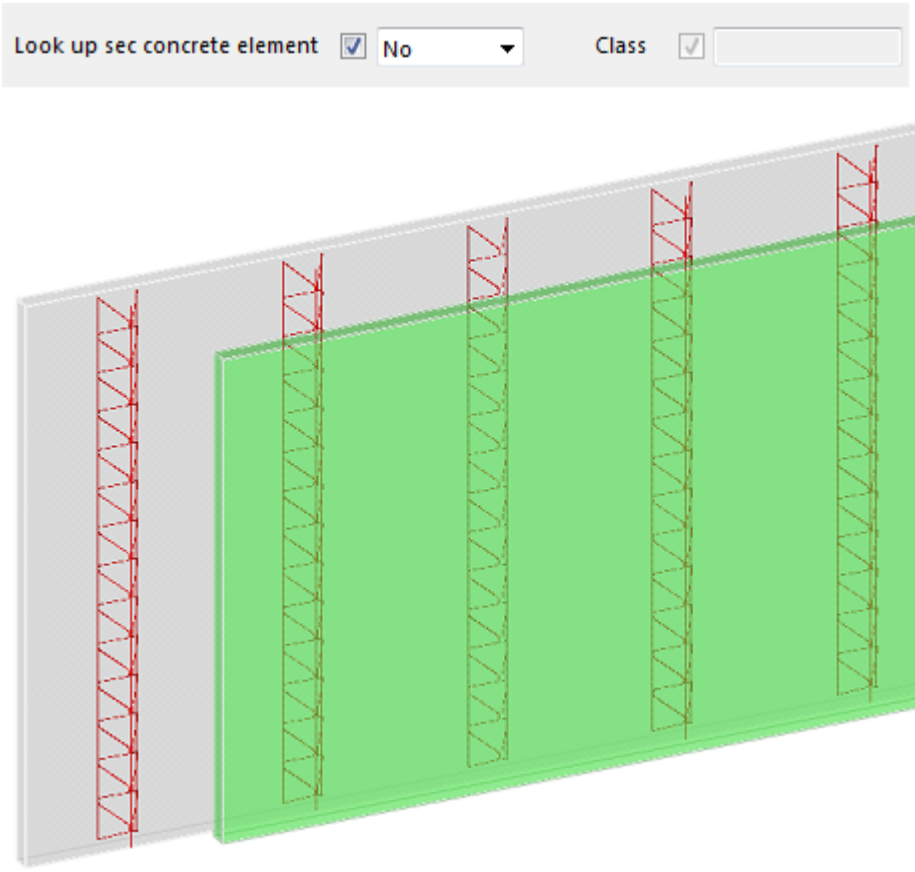
Look up sec concrete element

Select whether a second concrete element affects the creation of the girders. Define the class of the second wall in the **Class** box.

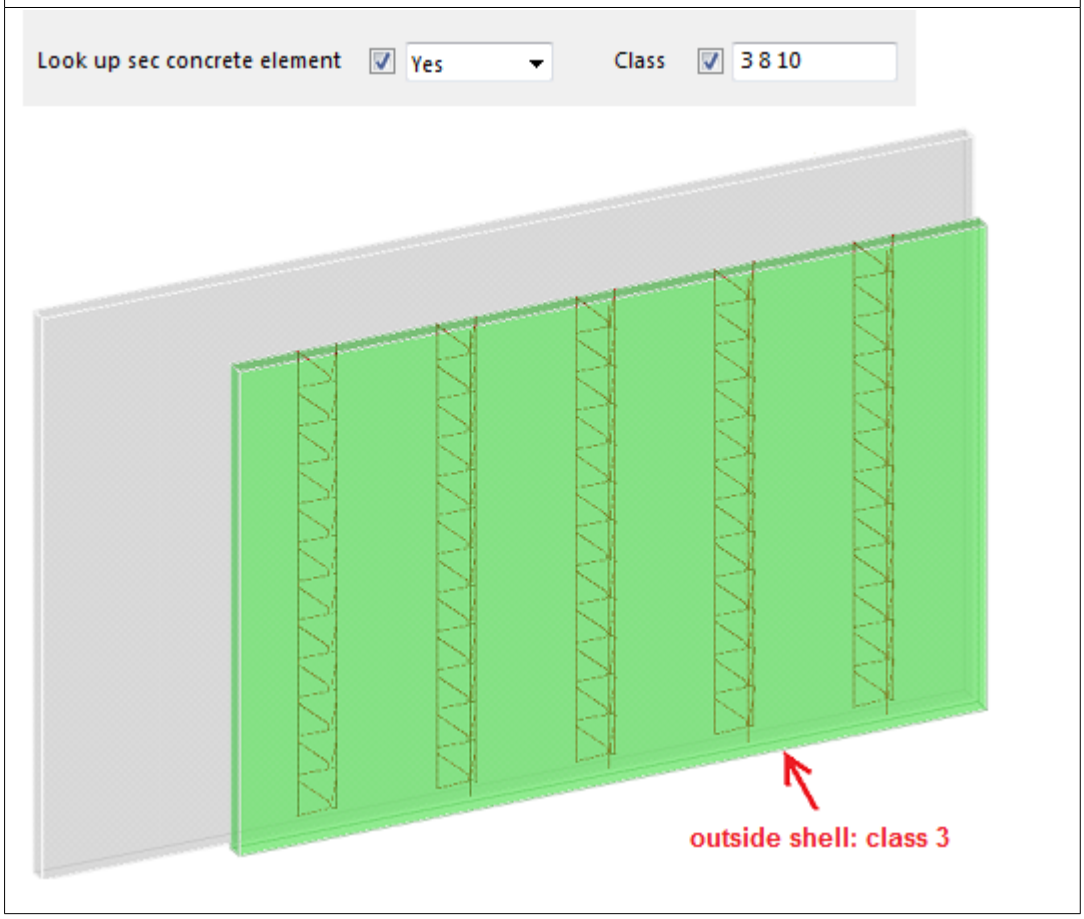
Select the first wall, and if the second wall matches the defined class, a girder is created. You can also enter a series of classes. You can use this creation method in combination with the options defined for openings on the **Geometry** tab.

The example below shows a hollow wall where inside and outside shells have different geometry.

Examples
A hollow core wall where inside and outside shells have different geometry. **Look up second concrete element** = **No**

Examples

A hollow core wall where inside and outside shells have different geometry. **Look up second concrete element** = **Yes**

Examples



UDA tab

You can define UDAs.

Braced girder (89)

Braced girder (89) creates braced girders between selected points in a precast concrete part, such as in a thin-shell slab or in a sandwich wall panel. The girders are cast into the concrete part and they also work as a part of the reinforcement and as a connector between the precast and cast-in-place concrete.

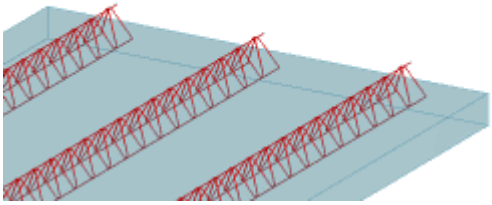
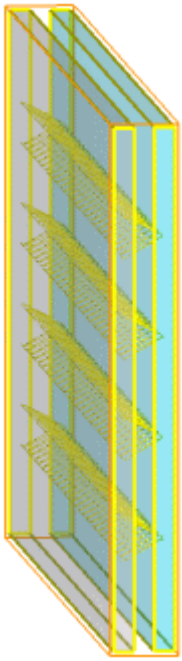
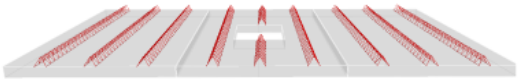
Objects created

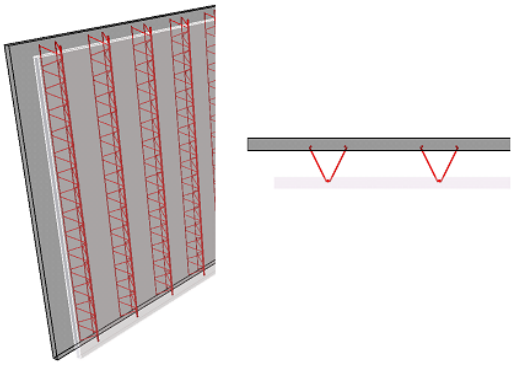
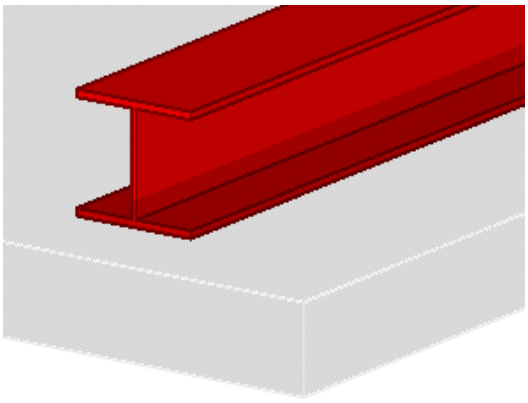
The girders consist of the following parts:

- Two bottom reinforcing bars
- One or two top reinforcing bars
- Two connecting reinforcing bars

Instead of using reinforcing bars, you can use profiles and plates to create the braced girders.

Use for

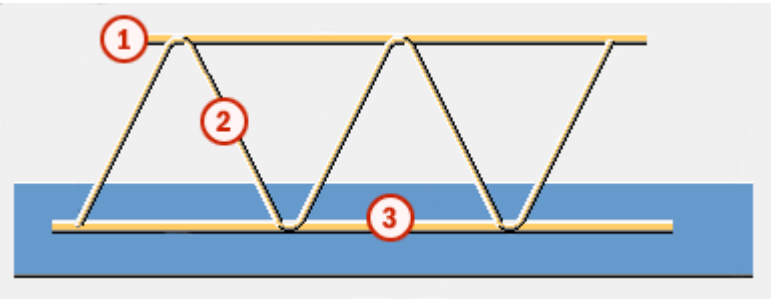
Situation	Description
	Braced girders are created in the precast concrete slab.
	Braced girders are created in the precast concrete sandwich panel.
	Braced girders are created in a wide plate floor with an opening.

Situation	Description
	Braced girders are created in a hollow wall.
	Profile as a girder.

Selection order

1. Select the concrete part.
2. Pick the starting point of a girder.
3. Pick the end point of a girder.

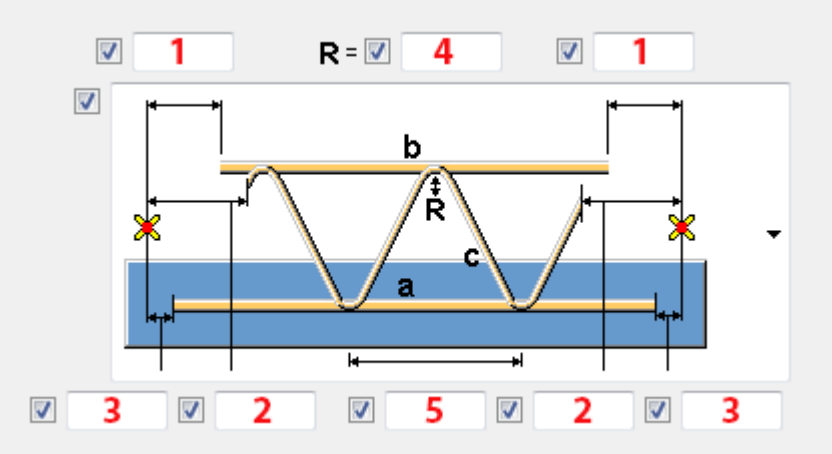
Part identification key



	Part
1	Top reinforcing bar
2	Connecting reinforcing bar
3	Bottom reinforcing bar

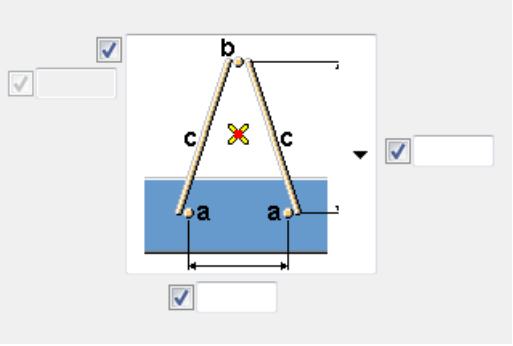
Picture tab
Use the **Picture** tab to control the creation, shape and dimensions of reinforcement bars.

Reinforcing bar dimensions and shape

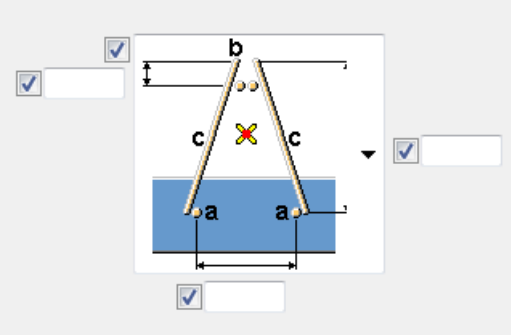


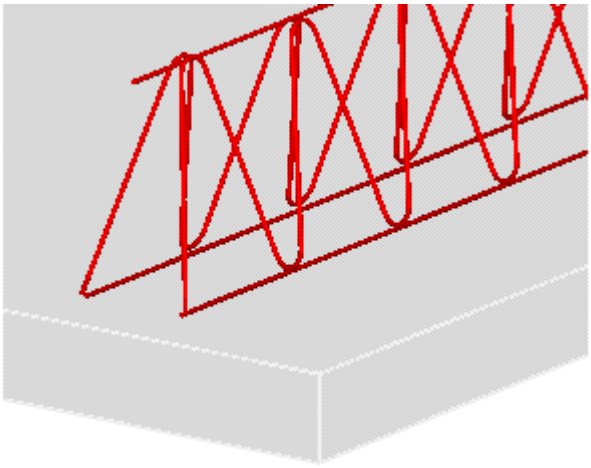
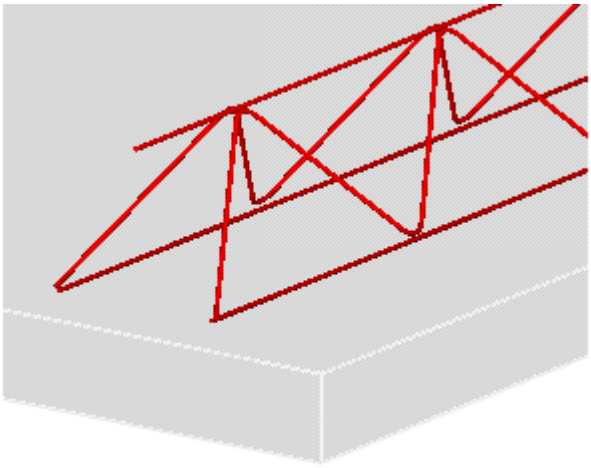
	Description
1	Distance between the end of the top reinforcing bar and the picked point.
2	Distance between the end of the connecting reinforcing bar and the picked point.
3	Distance between the end of the bottom reinforcing bar and the picked point.
4	Radius of the connecting reinforcing bar.
5	Distance between bendings.

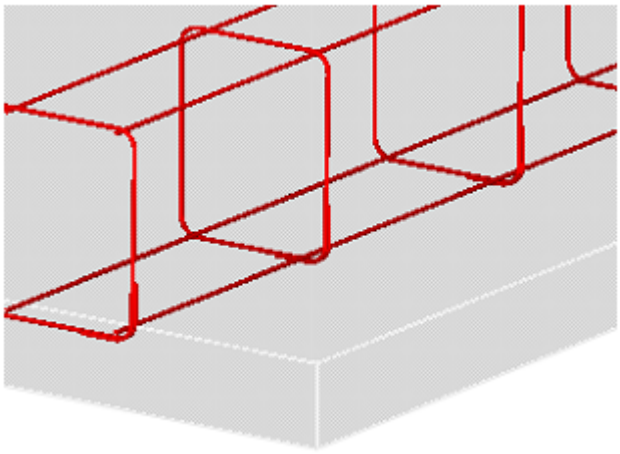
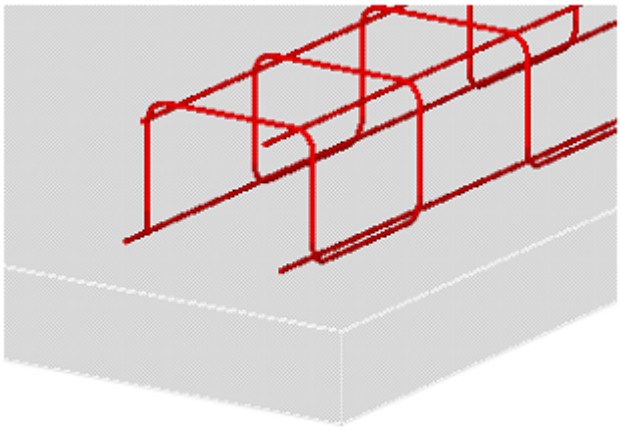
Define the distance between the bottom reinforcing bars, and the distance between the top and bottom reinforcing bars.



If you add two reinforcing bars at the top, you can define the distance of these reinforcing bars from the top of the connecting reinforcing bars.

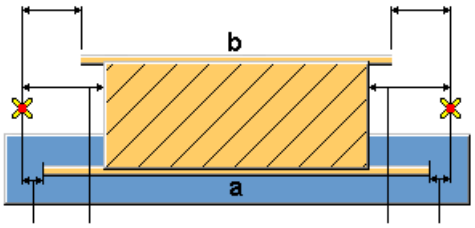


Examples



Examples



Girder creation

Option	Description
Create	Select the type of the bottom, top and connecting reinforcing bars (Bar a, b, c): • **Reinforcing bar** • **Steel rod** • **No** (The reinforcing bar is not created.)

Option	Description
Profile	Profile selection is activated when you select the following girder option: Define a prefix and a start number for the part position number, and material, name, comment, and class.
Plate	Plate creation is activated when you select the following girder option:  Define a prefix and a start number for the part position number, and material, name, comment, and class.
Add as	Select the method that is used to connect the girders to the cast unit: **Sub-assembly**, **Welded**, **No**

Reinforcing bar properties

Define the reinforcing bar properties for the top, bottom and the connecting reinforcing bars.

Option	Description
Size	Size of the reinforcing bar.
Grade/Material	Grade of the reinforcing bar.
Name **Prefix** **Start number** **Comment** **Class**	A name, a prefix and a start number for the part position number, and comment and class for the reinforcing bar.

Create as rebar assembly

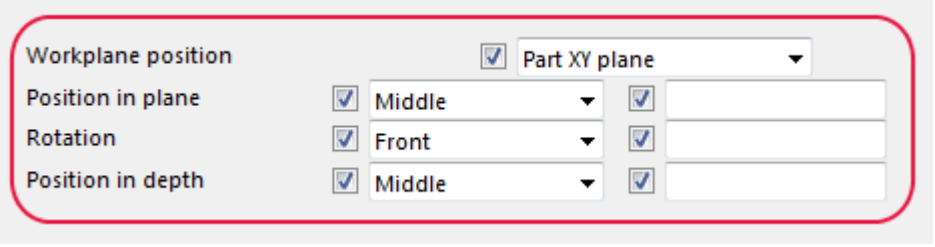
You can add the created reinforcement as a rebar assembly to the cast units with a predefined assembly type, name, prefix, and start number.

Option	Description
Create as rebar assembly	Select **Yes** to create all reinforcement as one rebar assembly, and to include it to the cast unit of the input part.
Add to existing rebar assembly	• **Do not add**: New rebar is added as a rebar assembly to the cast unit. • **As single bars**: Add all new rebars directly into an existing rebar assembly as rebars. • **As sub-assembly**: Add all new rebars to their own new rebar assembly, which is then included as a sub-assembly to the main rebar assembly.
Rebar assembly type	Select the rebar assembly type. If you do not select the type, the default value of the rebar assembly is used.
Name, **Profile**, **Start number**	Define the name, profile, and start number. If you do not define these, the default values of the rebar assembly are used.

Parts tab

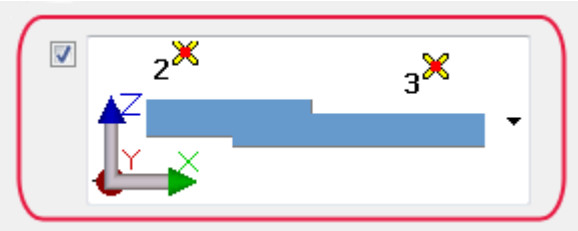
Use the **Parts** tab to control the positioning and number of girders.

Workplane orientation

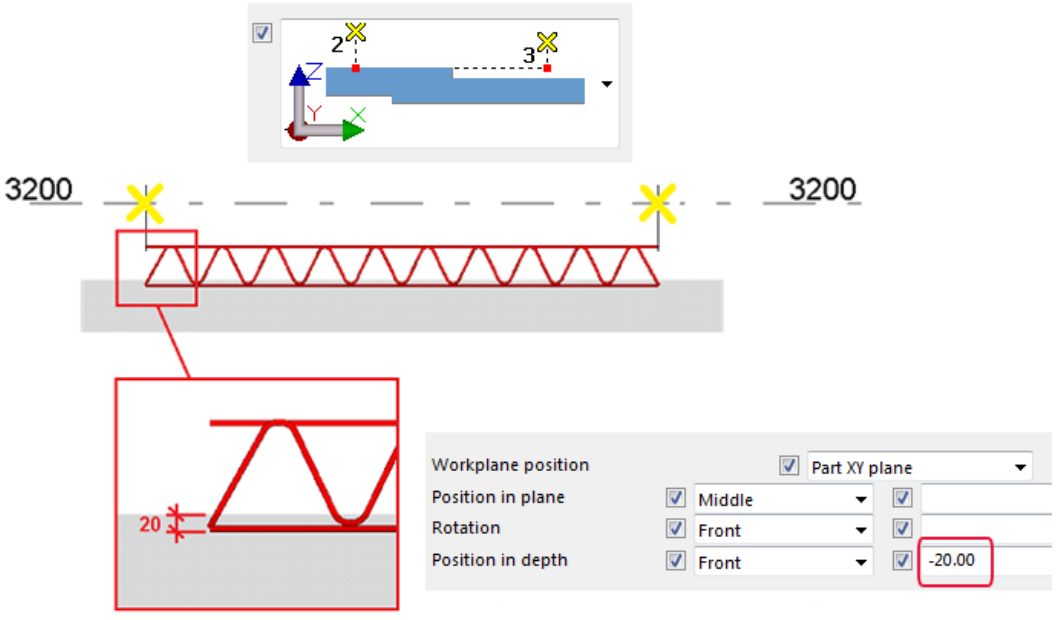


Option	Description
Workplane position	Use this setting to control to which side of the input part the girders are created. The **Model** option creates them according to the current work plane in the model. • **Part XY plane** • **Part YZ plane** • **Part ZX plane** • **Model**
	Use the **Position in plane**, **Rotation** and **Position in depth** options to define the orientation of the girder on the work plane.

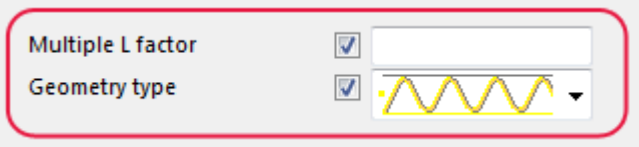
Girder position in z direction

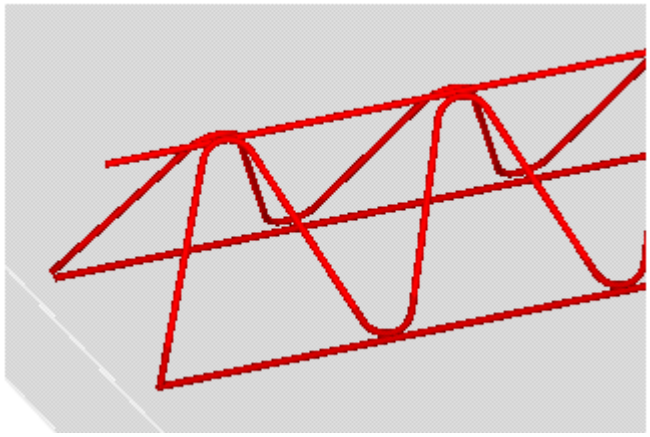
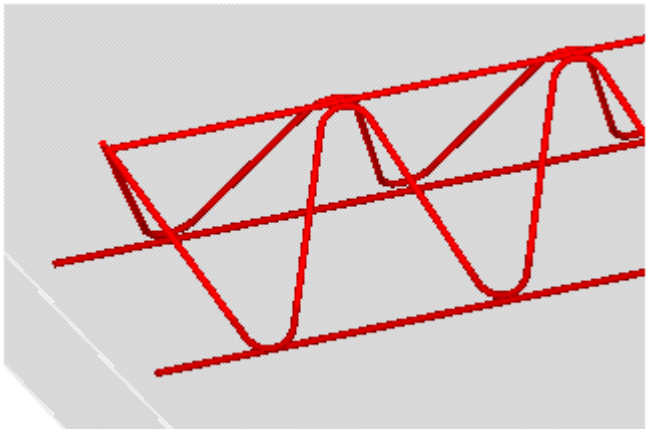


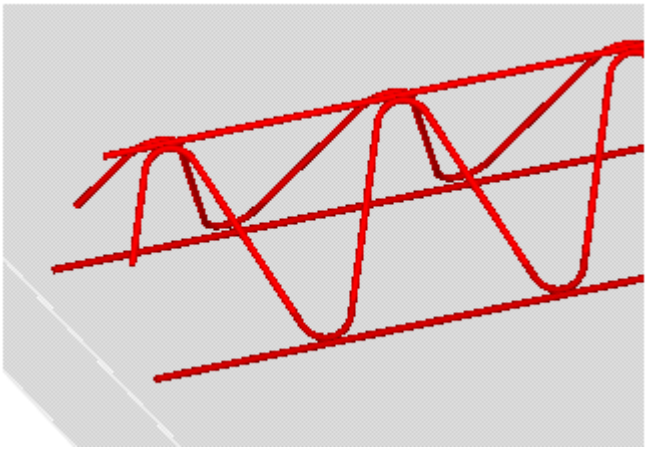
Select how the girders are positioned in the z direction.

Example
Braced girders are positioned to picked points. The braced girders are related to the top side of the part. 

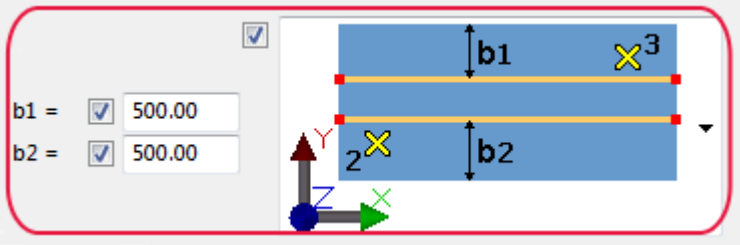
Geometry



Option	Description
Multiple L factor	Define the accuracy for the rounding of the girder length. The default value is 1.0. With the default value, there are no decimals in the girder length.
Geometry	Select the geometry for the connecting reinforcing bars. Examples:  

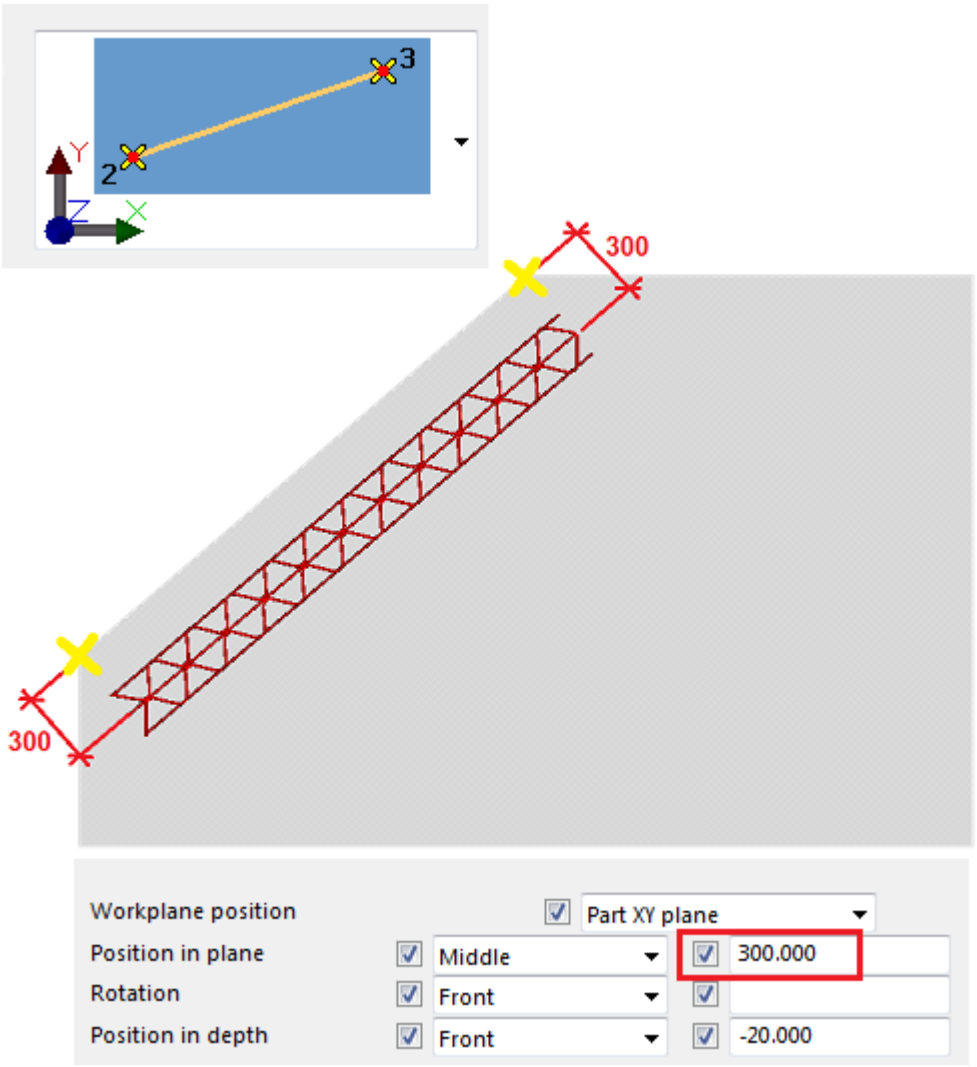
Option	Description
	

Girder position in y direction

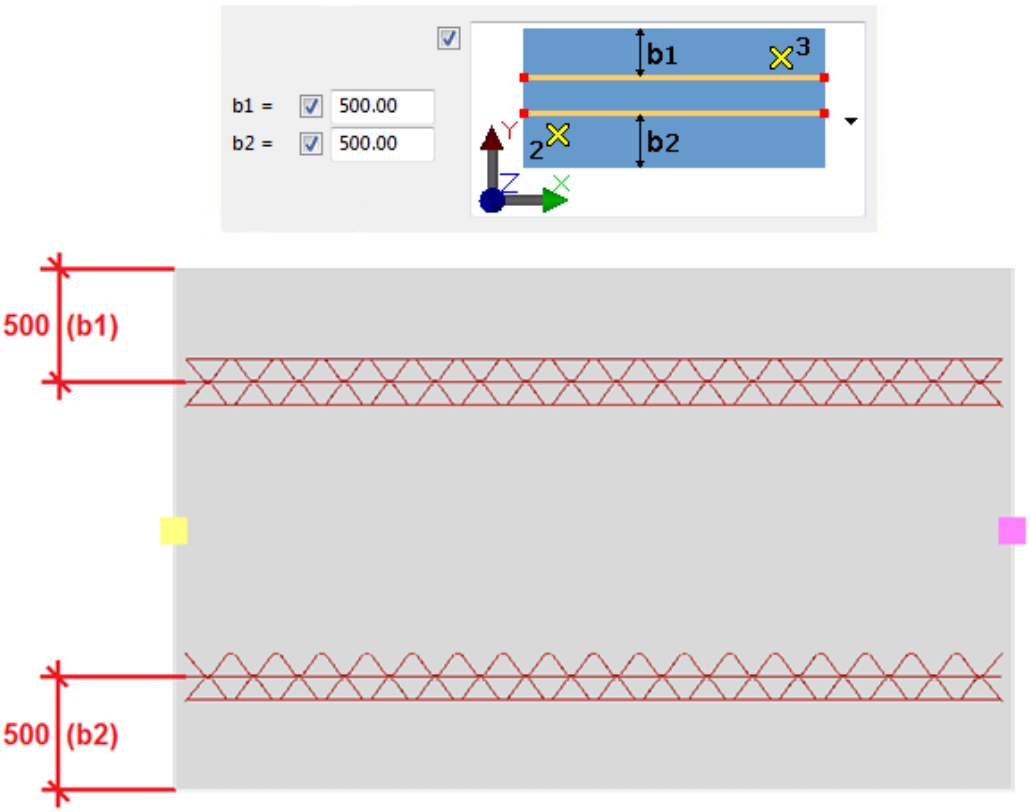


Select how the girders are positioned in the y direction.

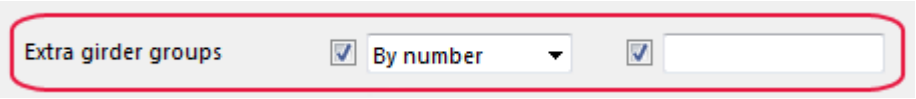
Examples


3
2
Y
Z
X
300
300
Workplane position
Part XY plane
Position in plane
Middle
300.000
Rotation
Front
Position in depth
Front
-20.000

Examples



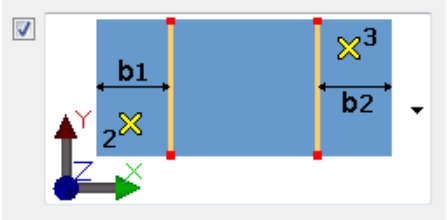
Extra girders



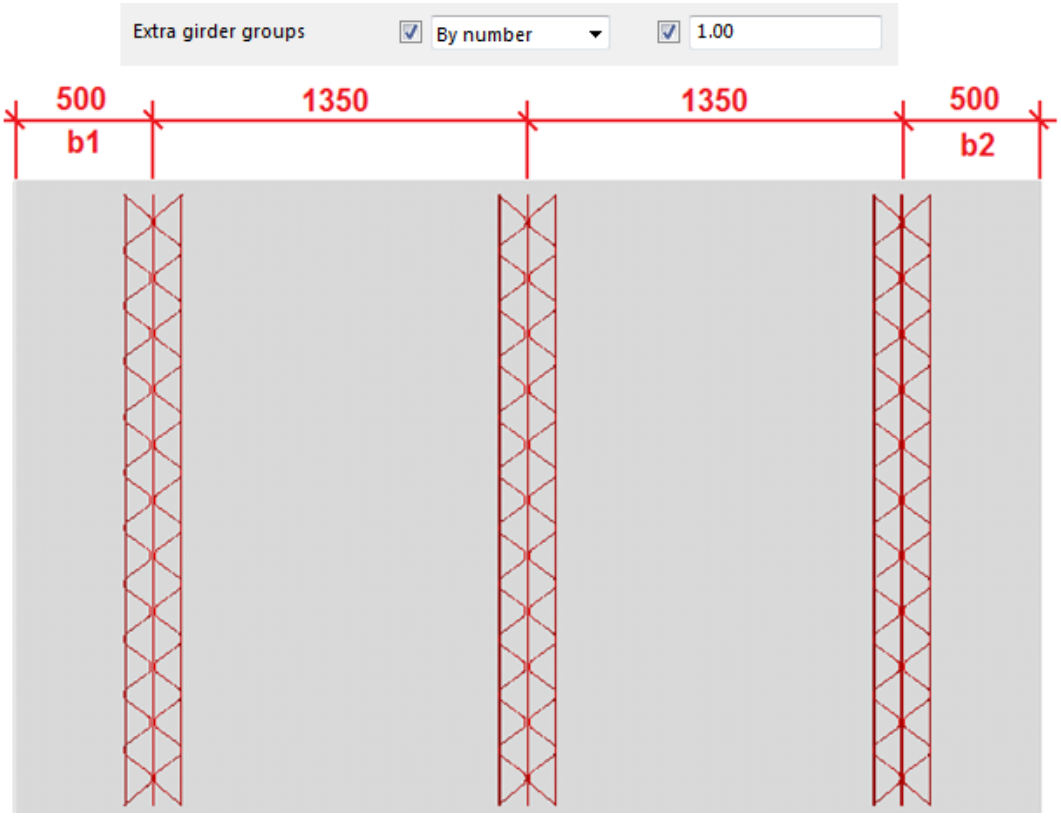
Select whether additional girder groups are created.

Examples

For example, if you have selected this option:

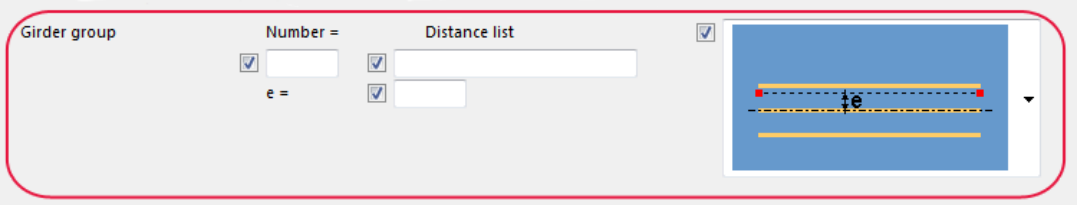


Examples
Extra girder groups is set to **None**: No additional girder groups are created between 2 existing braced girders.

Examples
Extra girder groups is set to **By number**: Additional girder groups are created based on the entered number. Distances between the groups are equally divided. 

Examples
Extra girder groups is set to **By distance**: The number of additional girder groups is based on the defined distance. The distances between the girder groups are equally divided.

Girder group

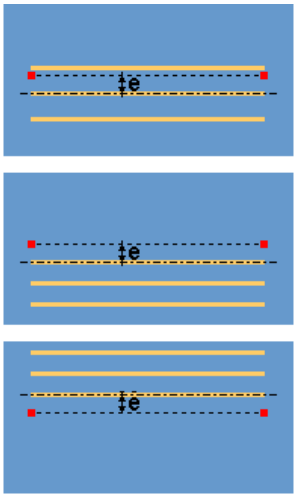


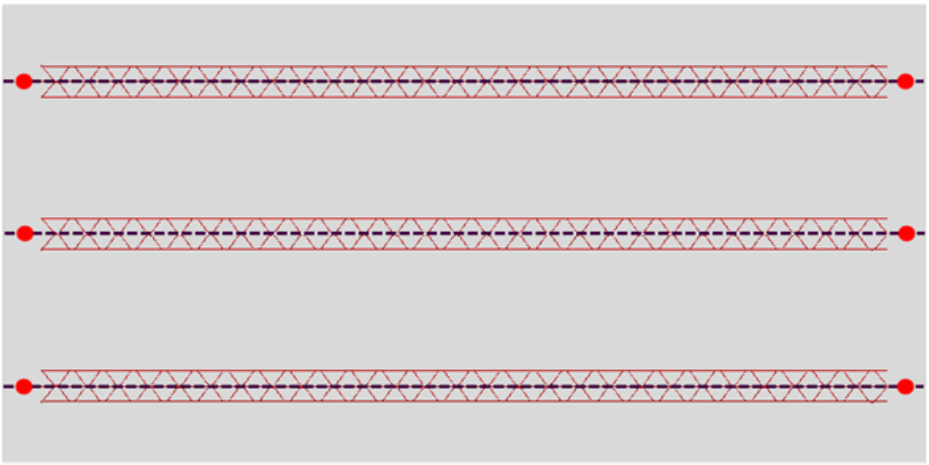
Define whether additional girders are created from the existing girders.

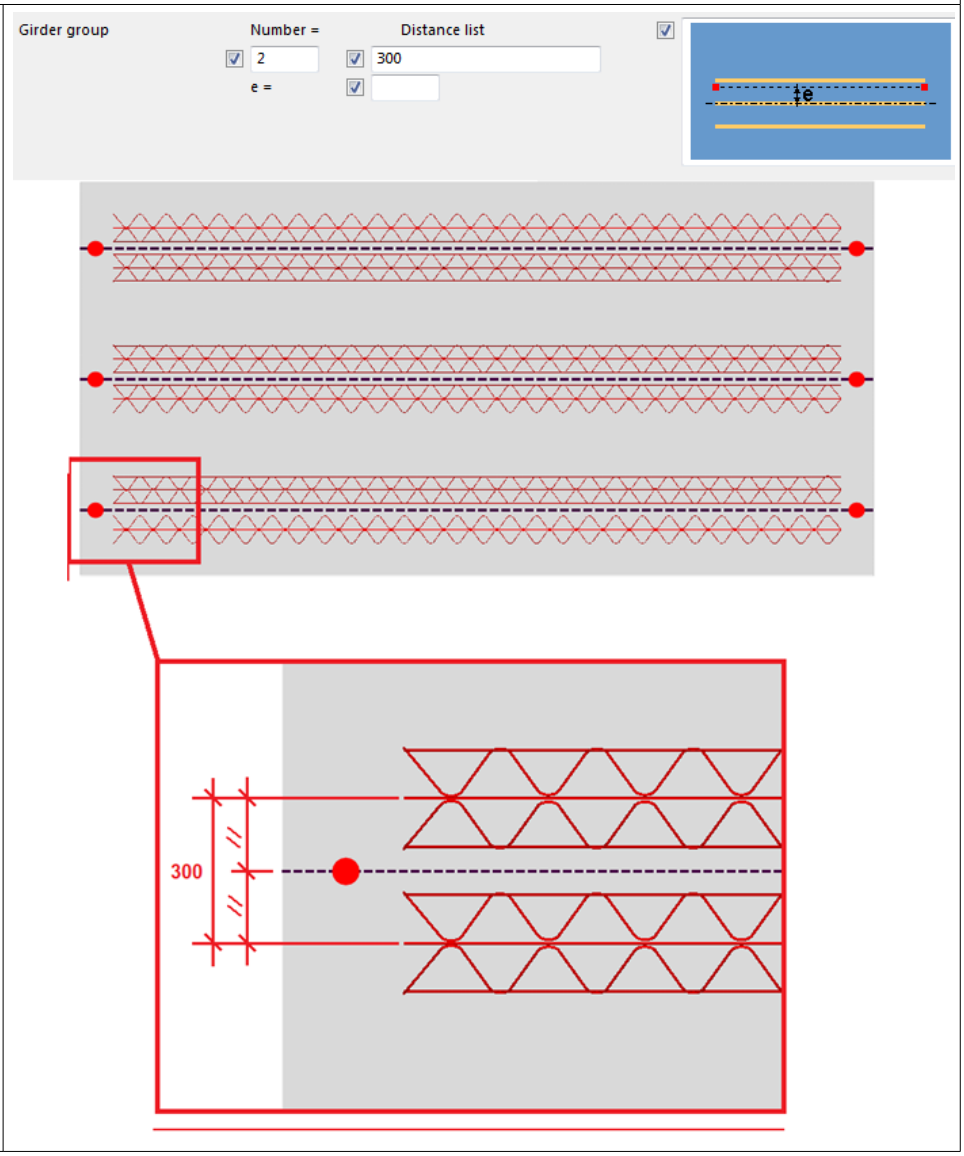
Number is the number of girders in the girder group.

Distance list is the distance between the girders in the girder group.

e = is the position of the girders from the reference line.

Option	Description
	• Girder group is positioned in the middle of the reference line. • Girder group is positioned on the right side of the reference line. • Girder group is positioned on the left side of the reference line.

Examples	
Existing girders	

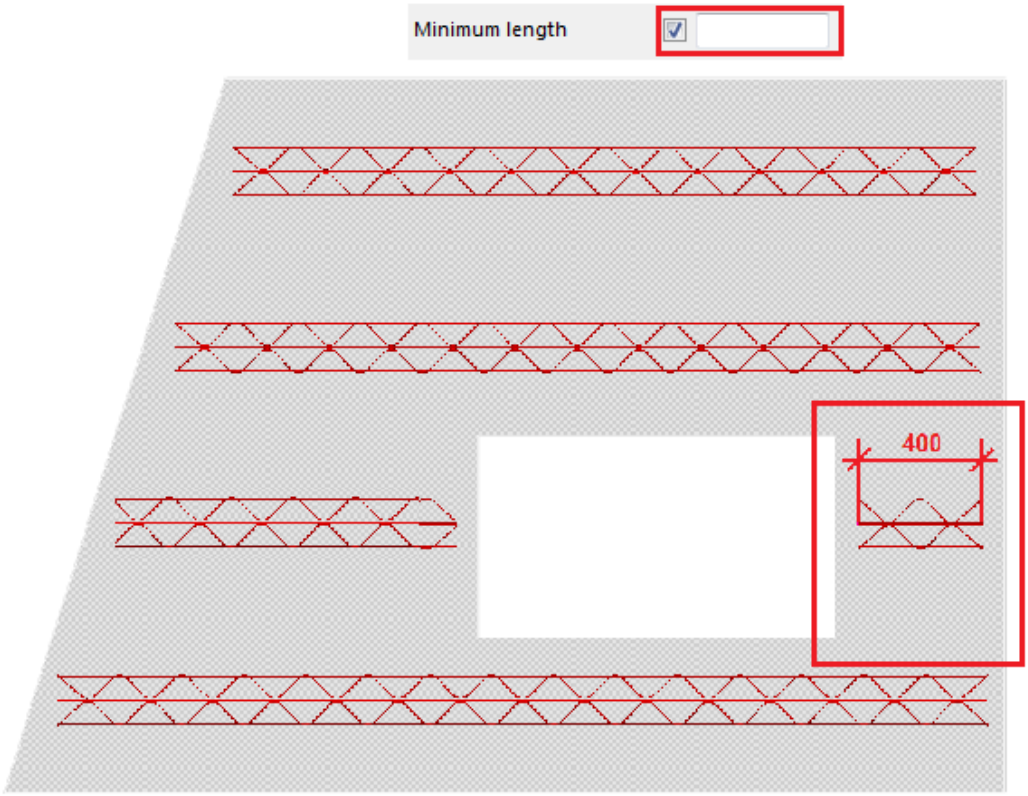
Examples	
Additional girders	

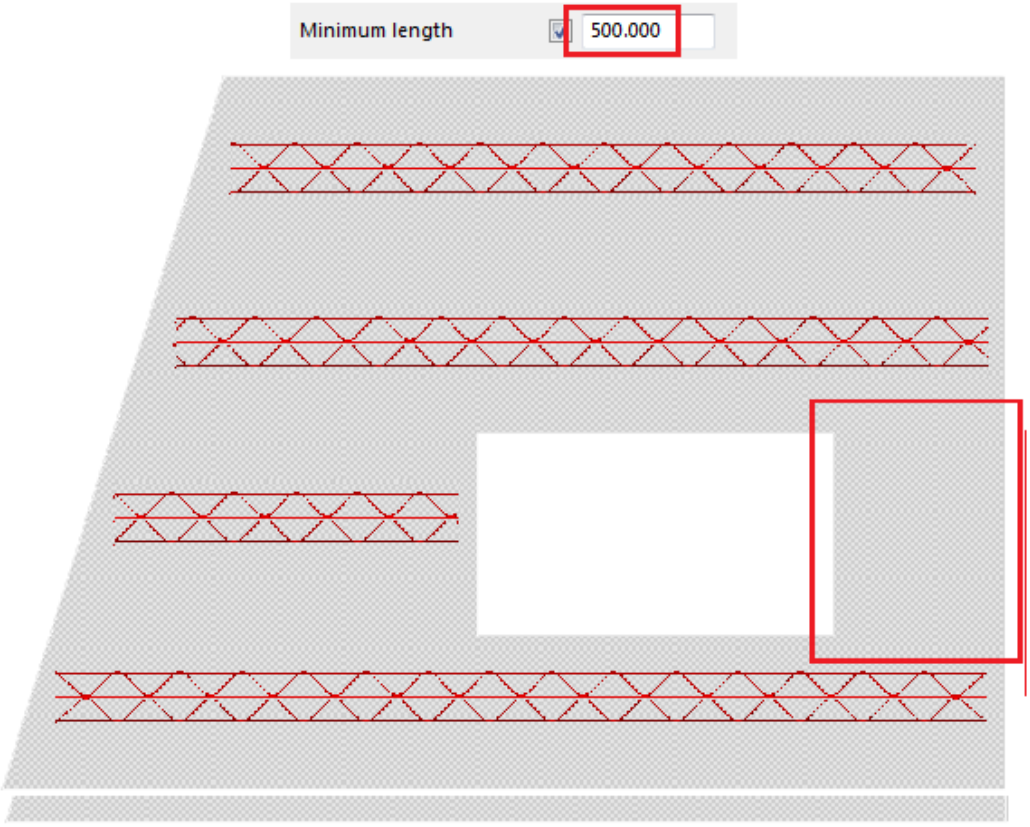
Geometry tab

Use the **Geometry** tab to control how openings and part length affect the girder creation.

Option	Description
Create always girders	Select whether girders are always created. If you select **Yes**, girders are created even when the girder is placed fully outside the concrete part.
Openings	Select whether girders are created in openings.

Option	Description
Length	Select how girders adapt to the part shape.
Minimum length	Define the minimum length of the girder.
Maximum length	Define the maximum length of the girder. The girder is split when the maximum length is reached.

Examples of minimum length
No minimum length defined: 

Examples of minimum length
Defined minimum length (no girder created): 

Double wall
Use the **Double wall** tab to select how a second concrete element affects the girders.

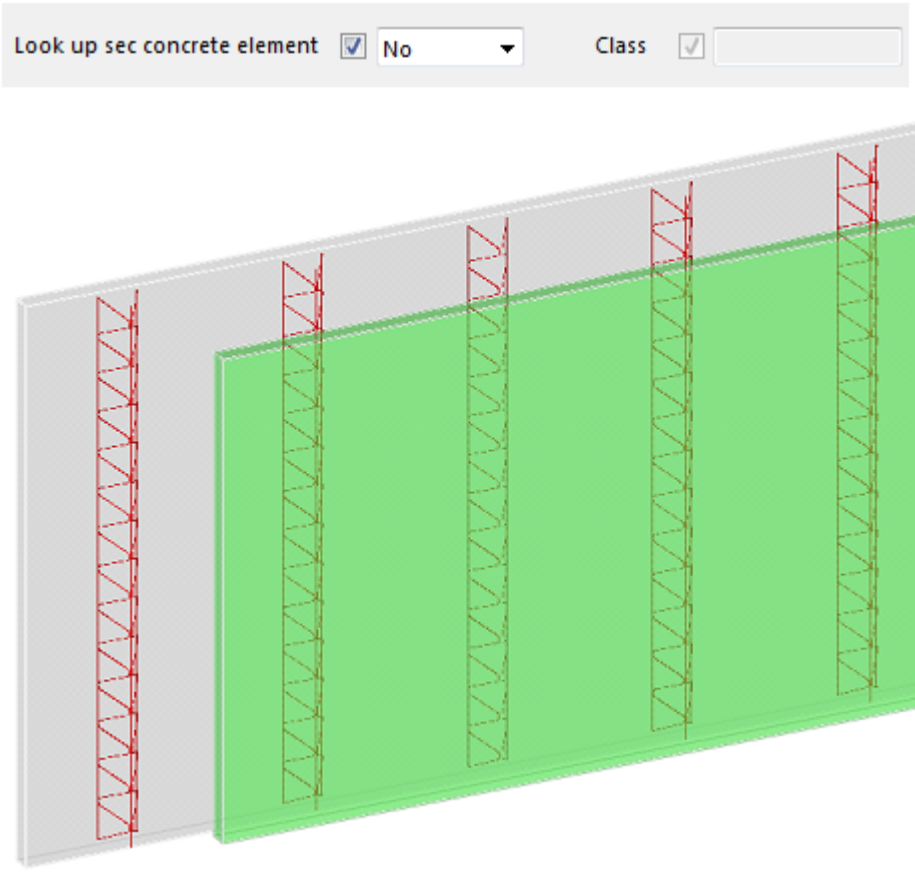
Look up sec concrete element

Select whether a second concrete element affects the creation of the girders. Define the class of the second wall in the **Class** box.

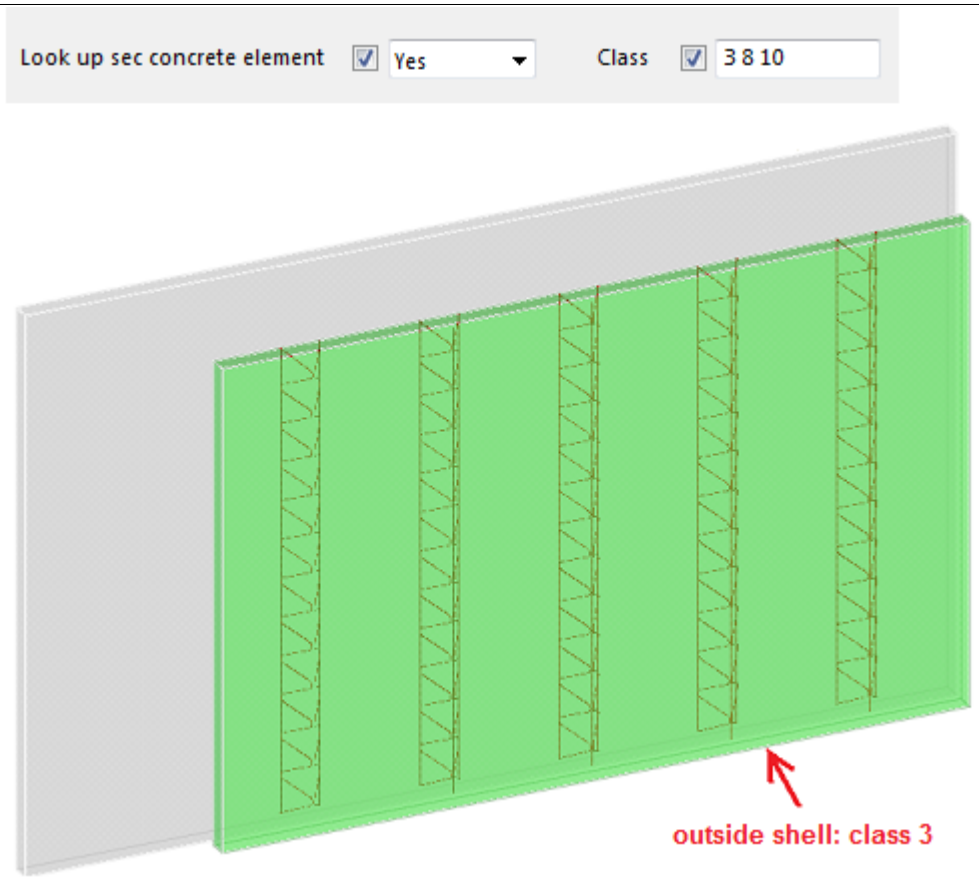
Select the first wall, and if the second wall matches the defined class, a girder is created. You can also enter a series of classes. You can use this creation method in combination with the options defined for openings on the **Geometry** tab.

The example below shows a hollow wall where inside and outside shells have different geometry.

Examples
A hollow core wall where inside and outside shells have different geometry. **Look up second concrete element** = **No**

Examples

A hollow core wall where inside and outside shells have different geometry. **Look up second concrete element** = **Yes**

Examples



UDA tab

You can define UDAs.

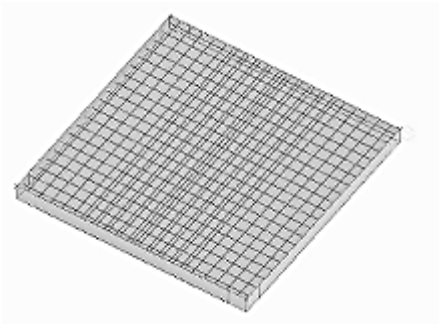
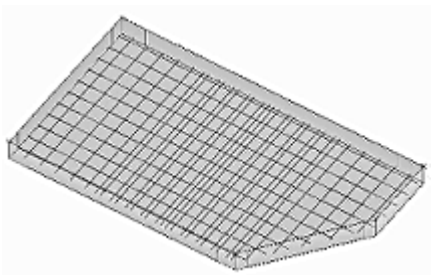
Reinforcement mesh array in area (89) / Reinforcement mesh array (91)

Reinforcement mesh array in area (89) / **Reinforcement mesh array (91)** create reinforcement meshes for an entire concrete part, or for a defined polygonal area. **Reinforcement mesh array (91)** is suitable especially for precast panels and slabs.

Objects created

- Reinforcement meshes

Use for

Situation	Description
	Array of overlapping rectangular meshes.
	Array of meshes in a polygonal area that you define. Meshes clipped to fit the defined area.

Before you start

- Create the concrete part.
- Set the work plane parallel to the plane where you want to create the mesh array.

Selection order

Component	Create reinforcement mesh
Reinforcement mesh array in area (89)	1. To attach the reinforcement mesh array: • to the entire part, pick the starting point of the mesh • to a selected area, pick points to define the polygonal shape of the mesh 2. Click the middle mouse button to create the reinforcement mesh array.
Reinforcement mesh array (91)	1. Select the part. The reinforcement mesh array is created automatically when the part is selected.

Picture tab

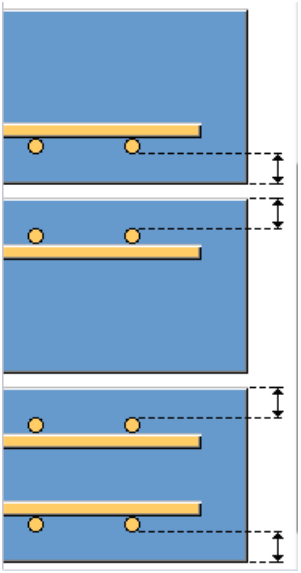
Use the **Picture** tab to define the mesh catalog type, cover thickness and location, mesh generation, cuts, and mesh area perimeter.

Mesh type

Select the mesh catalog type from the mesh catalog.

Cover thickness

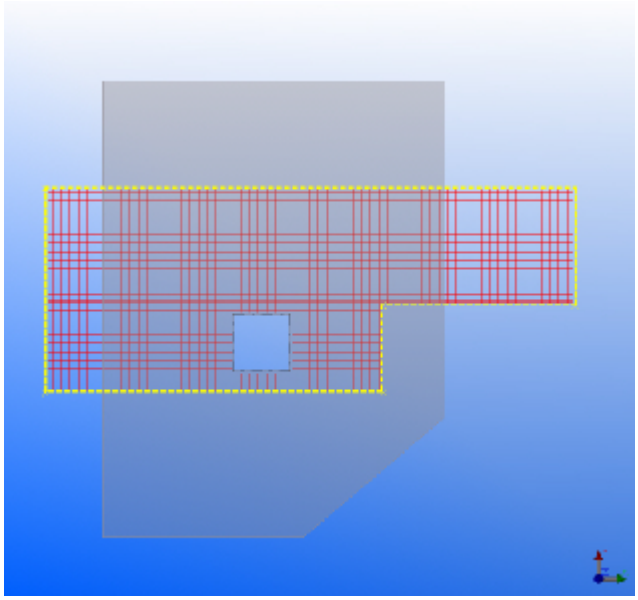
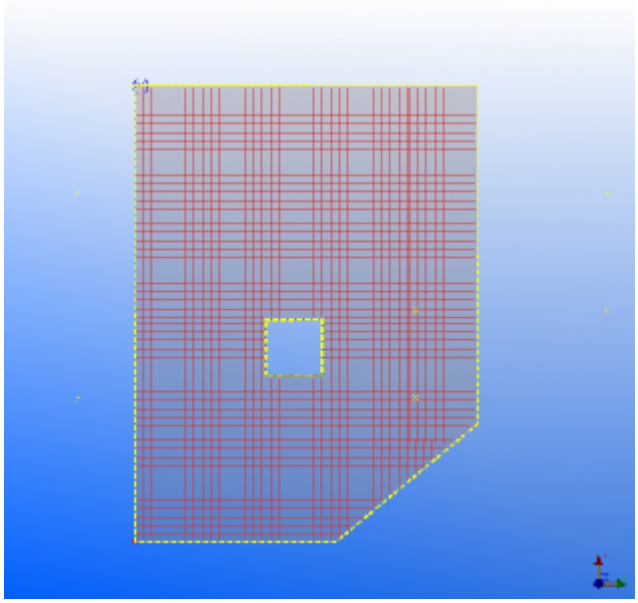
Define the mesh location and give values for **Cover thickness on plane** and **Cover thickness from plane**.

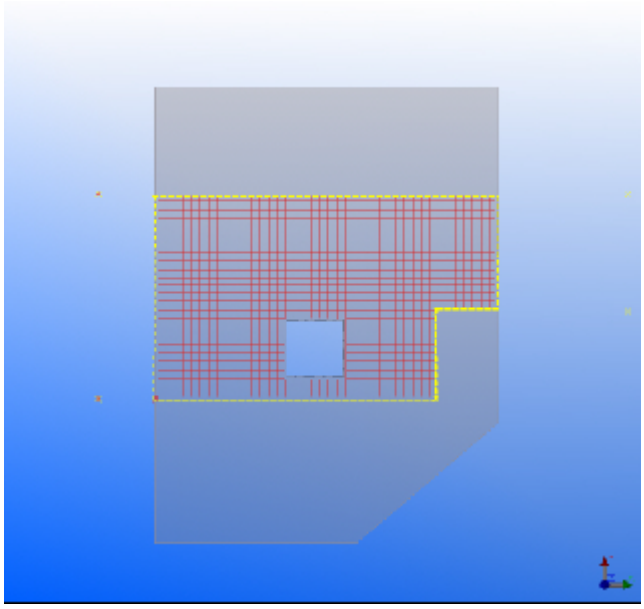


Define the cross bar location for top and bottom.

Mesh generation

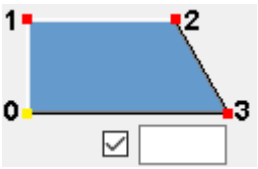
Option	Description
Generation direction	To rotate the mesh generation direction around the alignment point, enter an angle.
Direction of bottom layer	Select **Opposite of top layer** to mirror the bottom layer.
Cut by father part cuts	Select **Yes** to cut the mesh with the cuts made in the father part.
Clipping outside	Select **Yes** to clip the meshes to fit the part or the selected area.
Mesh area perimeter	Select the outline which the mesh follows. The meshes in the examples below are created by picking the same points but with different mesh area perimeters.

Option	Description
	Example: Polygon 
	Example: Part 

Option	Description
	Example: Polygon + Part 
Hole cover thickness	Define the hole cover thickness.

Generation start point

Enter the polygon generation start point index.



Overlapping tab

On the **Overlapping** tab, define the minimum and maximum overlapping along the longitudinal and crossing direction, and set the mesh offset.

Option	Description
Overlapping	Define the minimum and maximum overlap along the **Long side** and **Short side** of the mesh.
Mesh offset	Define how the meshes are placed. When you are creating both the top and the bottom mesh, you can shorten the starting mesh layer of both meshes separately. This allows you to move the whole mesh layer while the mesh layout defined in **Mesh offset** is retained.

Option	Description
Mesh start offset	Define mesh start offset for the first mesh. The length of the first mesh is divided by the selected value. For example, if you select the **Mesh start offset** to be **1/4**, the length of the first mesh is divided by **1/4**. If you select **Specified**, the **Start length offset** option becomes visible and you can enter the offset for the first mesh.

Attributes tab
On the **Attributes** tab, define the numbering properties, name, and class of the reinforcement meshes.

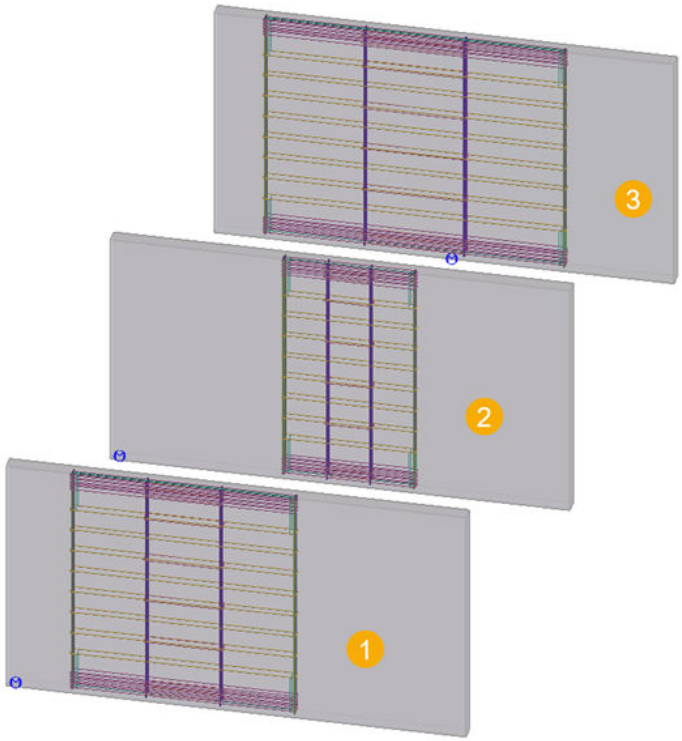
Rectangular area reinforcement (94)

Rectangular area reinforcement (94) creates reinforcement for a rectangular area. Select the part where the reinforcement is placed and pick two points to create the reinforcement. The part defines the thickness of the reinforced area, and the two points define the geometry and the location of the reinforced area.

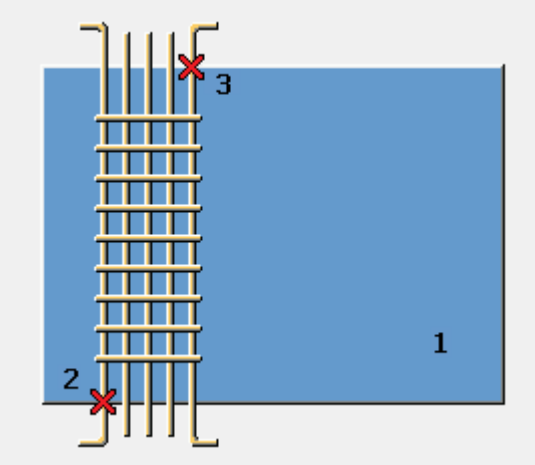
Objects created

- Longitudinal main bars: corner bars (4), side bars
- Stirrups
- Intermediate links
- Top and bottom bars

Use for

Situation	Description
	Concrete wall panel with corner and side bars inside the wall. Top and bottom ends reinforced. Intermediate links tie side bars at every second stirrup. **1** No rotation **2** Rotation in plane **3** Rotation out of plane

Selection order

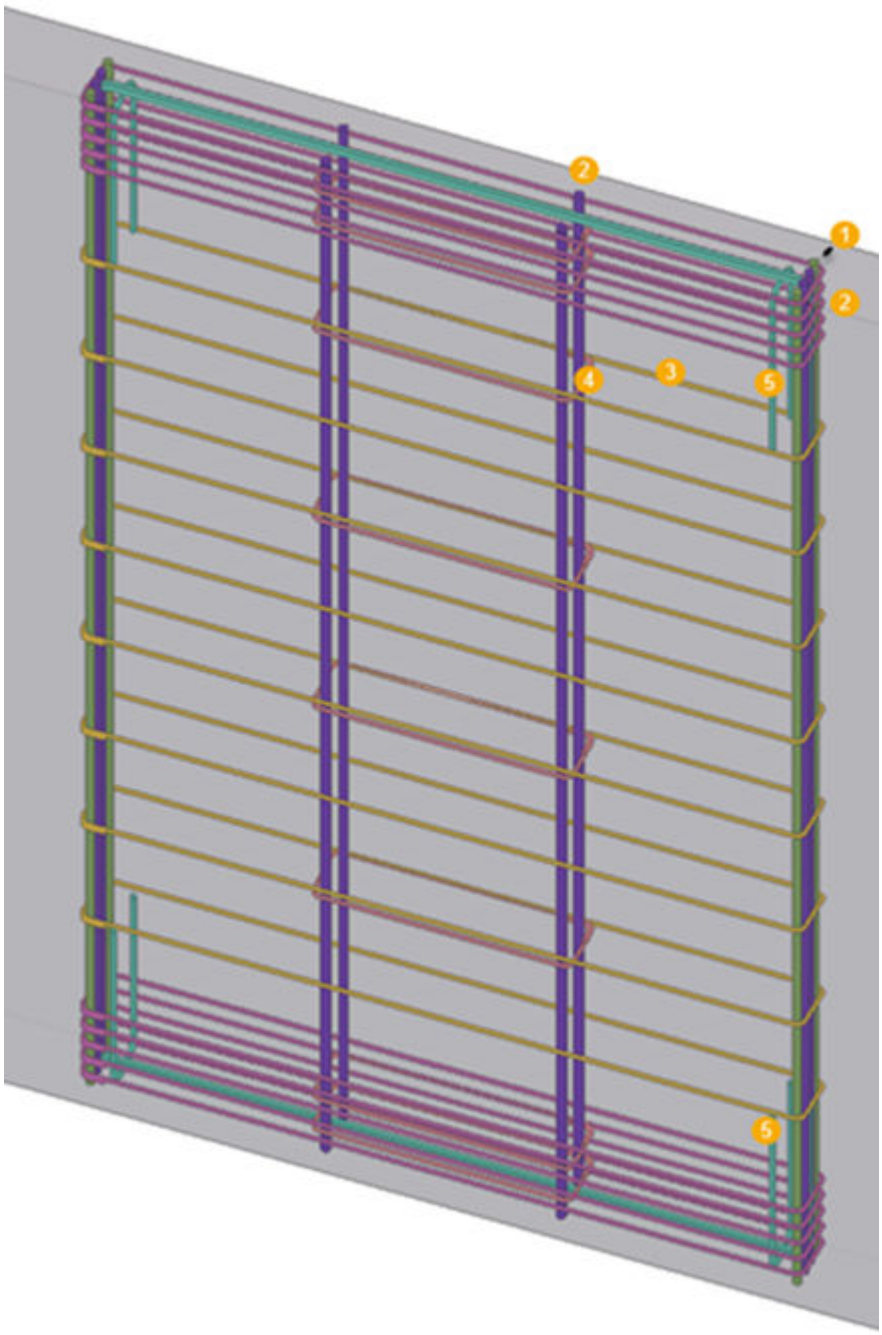


1. Select the part.
2. Pick the first point.
3. Pick the second point.

Picking the points (2 and 3) determines the area where the reinforcement is placed.

The reinforcement is created automatically when the second point is picked.

Part identification key



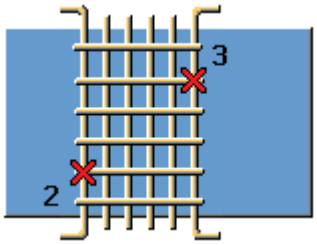
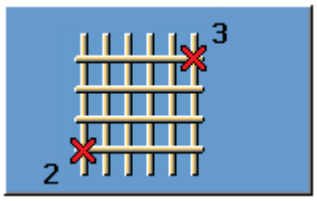
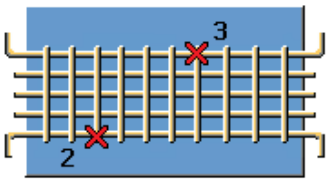
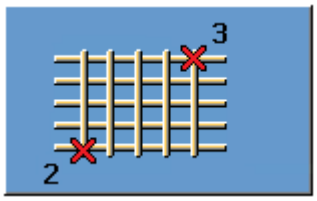
	Part
1	Corner bars
2	Side bars
3	Stirrups
4	Intermediate links

	Part
5	Top / Bottom bars

Picture tab
Use the **Picture** tab to select the reinforcement location and to set the reinforcement direction.

Rectangular area

Select how the reinforcement is extended over the wall.

Option	Description
	Column reinforcement area
	Area reinforcement with vertical rebars
	Beam reinforcement area
	Area reinforcement with horizontal rebars

Rotation

Rectangular area reinforcement (94) is based on the geometry of a standard panel. Using the component on slabs or beams in certain planes can affect the geometry of the reinforcement. By using the **In plane** or **Out of plane** options, you can adjust the reinforcement to the correct behavior.

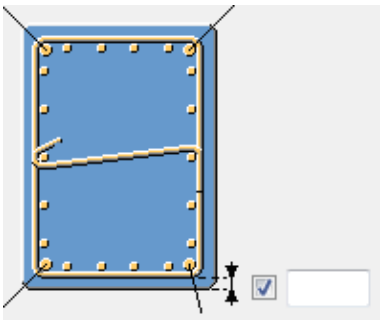
Select the reinforcement direction in different part geometry and rotation situations:

- **No**
- **In plane**
- **Out of plane**

Main bars tab
Use the **Main bars** tab to control the corner bar properties, the symmetry options, rotation, and concrete cover thickness.

Basic corner bar properties

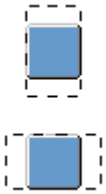
Define the grade, size and bending radius of the corner bars. The active settings depend on the selected symmetry option.

Option	Description
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Bending radius	Define the bending radius of the corner bars.
	Define the concrete cover thickness. Select whether the thickness is the same on all sides.

Symmetry options

Select the symmetry option. Use the symmetrical conditions to define which of the corner bars have the same grade, size, and bending radius properties. The corner bars that have same properties are symmetrical.

Rotation



In square reinforced parts, you can select the perpendicular sides if the sides require different reinforcement. You can rotate all reinforcement by 90 degrees.

Additional corner bar properties

Option	Description
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Name	Define a name for the main bars. Tekla Structures uses the name in drawings and reports.
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.

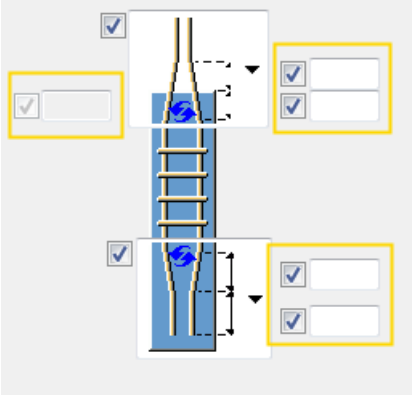
Hooks at top / bottom

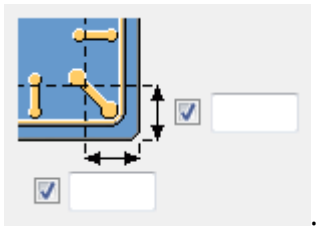
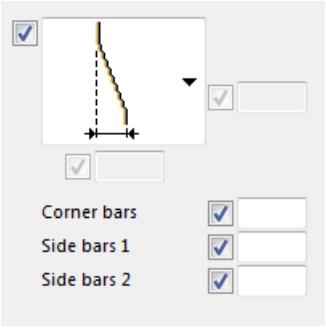
Option	Description
Type	Select the hook angle.
Length	Length of the hook.
Bending radius	Bending radius of the hook.

Bar ends tab

Use the **Bar ends** tab to control the length of the vertical and horizontal extensions for the corner bars and side bars, and the cranking.

Option	Description
Bar creation	Select whether the extensions of corner bars and side bars are created symmetrically on both sides of the rectangular reinforcement. If you select **Not symmetrical**, you can enter separate extension values for the opposite sides of the reinforcement.

Option	Description
Vertical extension	Define the length of the vertical extension for corner bars and side bars outside the reinforced part. The active settings depend on **Bar creation**: **Symmetrical** or **Not symmetrical**.
Horizontal extension	Define the length of the horizontal extension for corner bars and side bars. The active settings depend on **Bar creation**: **Symmetrical** or **Not symmetrical**.
Top corner bar direction **Bottom corner bar direction**	Select the direction of the corner bars.
Cranking	You can create cranked reinforcing bars at the top and at the bottom of the reinforcement. To activate the cranking options, select in the **Top corner bar direction** and **Bottom corner bar direction** lists. Define the dimensions for the cranked bars.  To successfully create the cranked reinforcing bars, ensure that the bending radius is not too large.

Option	Description
Edge dimensions	Define the edge distance from the reinforced part corner to the point where the cranking starts  . Define the cranking separately for the corner bars and the side bars. 

Side bars tab

Use the **Side bars** tab to control the number of side bars, side bar spacing and placing, symmetry options, and properties.

Number of side bars

Define the number and spacing of side bars. You can define two sets of side bars on each side of the rectangular reinforcement.

You can define the side bars separately for each side.

The active settings depend on the selected symmetry option.

Placing of side bars

Select the horizontal and vertical placing for the side bars. Select whether the bars are placed starting from the corner or at equal spaces.

	Distance between corner bars and side bars.
	Distance between side bars.

Symmetry options

Select the symmetry option. Using the symmetrical conditions you can define which side bars are symmetrical, and which side bars use the same properties.

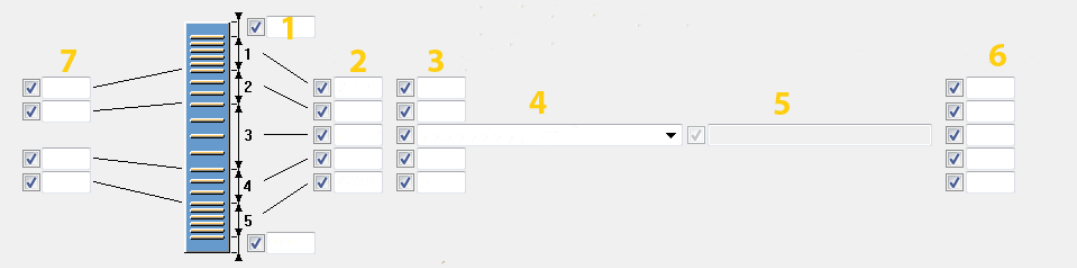
Side bar properties

Option	Description
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Bending radius	Define the bending radius of the side bars.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Name	Define a name for the side bars. Tekla Structures uses the name in drawings and reports.
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.

Stirrups tab

Use the **Stirrups** tab to control the stirrup properties.

Stirrup dimensions



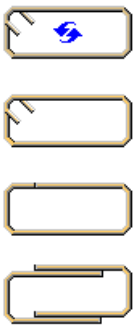
	Description
1	Define the thickness of the concrete cover over the stirrups at the top and the bottom of the rectangular reinforcement. The default cover thickness is 50 mm. Group **1** is the top stirrup group, **5** is the bottom stirrup group. Group **3** is always created.
2	Define the spacing of stirrups in each stirrup group.
3	Define the number of stirrups in each stirrup group.
4	Select how the stirrups are distributed.
5	If you select the **Distance list** option, enter different spacing values for the groups.
6	Define the cover thickness for each stirrup group.
7	Define the gaps between the stirrup groups.

Create stirrups

Select whether the stirrups are created as individual reinforcing bars, rebar group, or spiral rebar group.

Rebar lapping at stirrup corners

Select how the rebars lap at the stirrup corners. The options are 135-degree hooks or 90-degree hooks at the bar end, or overlapping U-shape stirrups.



You can define the overlap length for the U-shape stirrups.

Ignore cuts

If you have a recess or a hole in the reinforced part, you can select to ignore the cuts at the top and the bottom of the part when the stirrups are created.

Stirrup properties

Option	Description
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Name	Define a name for the stirrups. Tekla Structures uses the name in drawings and reports.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.
Overlap length	Define the overlap length for the U-shape stirrups.

Intermediate links tab

Use the **Intermediate links** tab to control the intermediate links to tie all side bars.

Intermediate links are created for each stirrup group.

NOTE Intermediate links are created between **Side bars 1** or **Side bars 2** that are symmetrical.

For **Side bars 2** intermediate links are created only if no **Side bars 1** are created.

Create as one group

Select whether all the stirrups are created as a single group.

Properties of intermediate links

Option	Description
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Name	Define a name for the stirrups. Tekla Structures uses the name in drawings and reports.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Prefix	Define a prefix for the part position number.

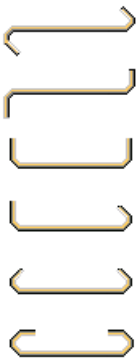
Option	Description
Start number	Define a start number for the part position number.

Spacing of intermediate links

Option	Description
	Spacing is the same as stirrup spacing.
	Spacing is double the stirrup spacing (intermediate links at every second stirrup).
	Same as above, but for alternate stirrups.
	No intermediate links are created.

Intermediate link type

Select the type of the link.



Intermediate link pattern

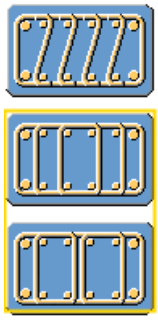
Select whether intermediate links go in one direction or in a crossing pattern.

Hook orientation

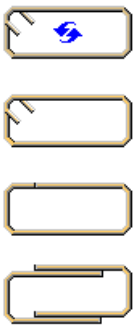
Select the hook orientation for both sides.

Closed stirrups as intermediate links

Select whether to use closed stirrups as intermediate links.



If you select closed stirrups, select the type of the bar lapping at the stirrup corners.



Zone without intermediate links

Define the zone length where intermediate links are not created. In this zone, the stirrups tie the side bars. The distance is measured from the stirrup corner.

If you have selected to use closed stirrups, you can define the distance from the corner of the closed stirrups.

Top / Bottom tabs

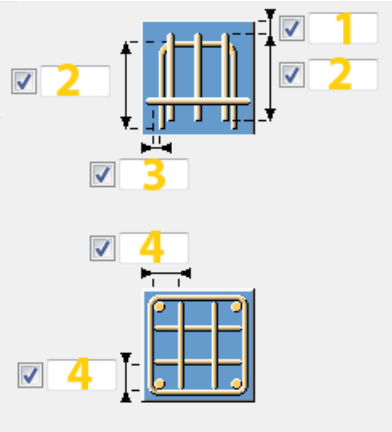
Use the **Top** and **Bottom** tabs to control the top and bottom reinforcement thickness of the concrete cover, number of bars, spacing and rotation.

Number and spacing of bars

Define the number and spacing of the top or bottom bars.

You can define the number of transverse bars in both cross directions.

Top / bottom reinforcement



	Description
1	Define the concrete cover thickness from the top / bottom of the rectangular reinforcement.
2	Define the length of the rebar legs.
3	Define the concrete cover thickness from the sides of the rectangular reinforcement.
4	Define the distance from the edge of the rectangular reinforcement to the edge of the first rebar in the group.

Placing of top or bottom bars

Select the horizontal and vertical placing for the top or bottom bars. Select whether the bars are placed starting from the corner or at equal spaces.

Option	Description
	Distance between corner bars and side bars.
	Distance between side bars.

Rotation

Select how to rotate the reinforcement at the top or bottom.

Option	Description
	No rotation. Transverse bars are perpendicular to the longer side of the rectangular reinforcement.
	Rotation angle is 90 degrees. Transverse bars are parallel to the longer side of the rectangular reinforcement.

Top and bottom reinforcement properties

Option	Description
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Name	Define a name for the top or bottom bars. Tekla Structures uses the name in drawings and reports.
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.

Rebar assembly tab

Use the **Rebar assembly** tab to add the created reinforcement as a rebar assembly to the cast units.

Create as rebar assembly

You can add the created reinforcement as a rebar assembly to the cast units with a predefined assembly type, name, prefix, and start number.

Option	Description
Create as rebar assembly	Select **Yes** to create all reinforcement as one rebar assembly, and to include it to the cast unit of the input part.
Rebar assembly type	Select the rebar assembly type. If you do not select the type, the default value of the rebar assembly is used.
Name, **Profile**, **Start number**	Define the name, profile, and start number. If you do not define these, the default values of the rebar assembly are used.

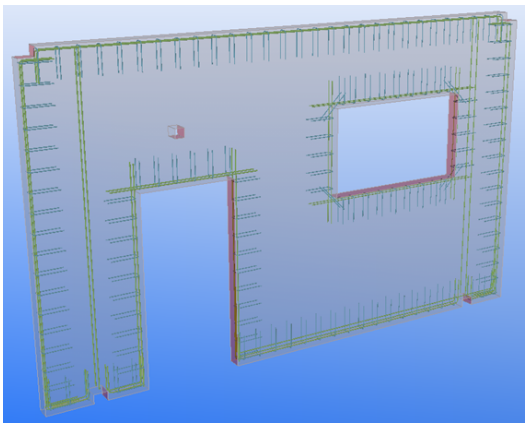
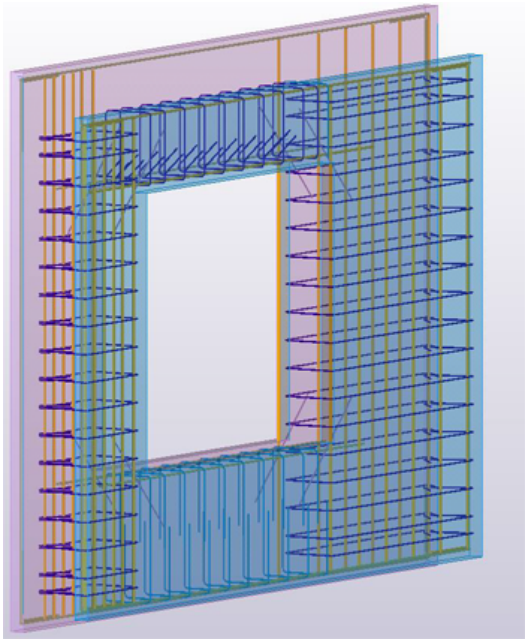
Wall panel reinforcement / Double wall edge and opening reinforcement

Wall panel reinforcement is used for detailing single precast wall panels. It includes the reinforcement for the panel edges, around the openings, a mesh for the whole panel and additional bars, for example, with a wall shoe connection. **Double wall edge and opening reinforcement** is used for detailing double walls by connecting two panels to each other. It includes the reinforcement for the panel edges and around the openings.

Objects created

- Reinforcement
- Stirrups
- Mesh (for single panels)

Use for

Situation	Description
	Wall panel reinforcement with openings and reinforcement in the panel edges and around openings.
	Double wall with opening and reinforcement around the opening. Ring reinforcement goes around both shells. All reinforcing bars that connect the shells (U-bars and stirrups) are attached to the shell that is selected at creation. Ring, beam and column reinforcing bars are attached to the selected shell that has been the input.

Selection order

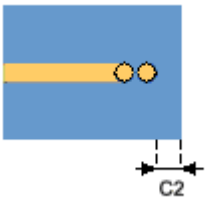
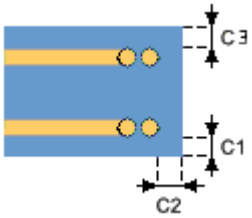
1. Select the concrete part to reinforce.

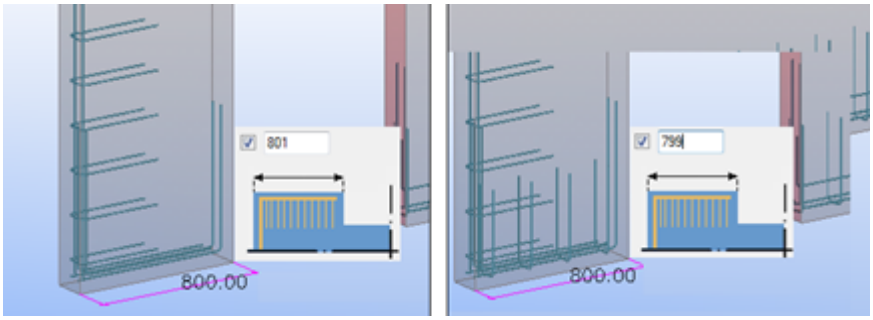
   The reinforcement is created automatically.

Picture tab

Use the **Picture** tab to select how the corners of a wall panel are reinforced.

Option	Description	
Corner reinforcement	Select how the corners are reinforced and the bar is offset on both sides of the corner.	

Option	Description	
Minimum breaking distance	Cuts smaller than this value are ignored and the reinforcing bars are not cut.	
Rebar count		Creates a single layer of reinforcement.
		Creates two layers of reinforcement.
Cover thickness	Distance from the edge of the wall panel to the side of the outermost bar.	
Door minimum height	Cuts greater than this value are considered as door openings and the additional reinforcing bars that are needed, for example, in shoe connections are not created.	
Minimum bar length	Edge bars shorter than this value are not created.	
Create stirrups priority	Select whether horizontal or vertical reinforcing bars are on top.	

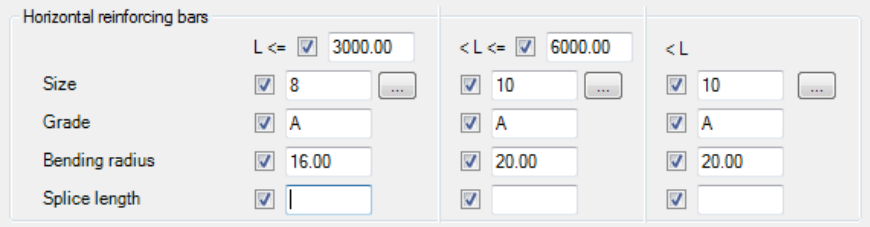
Option	Description
Minimum edge length	The edge stirrups are not created if the dimension is less than the defined minimum value. 
Create reinforcement in	Select whether the reinforcement is created in the selected part or in the entire cast unit. This setting is available in **Wall panel reinforcement**.
Longitudinal overhangs	Select **Target** to create variable overhang lengths. This creates meshes as full length meshes minus the cover thickness. Select **Exact** to create centered meshes with a fixed overhang length.
Minimum gap	Define the gap between the meshes in the corners.

Reinforcement tab

Use the **Reinforcement** tab to create the reinforcing bars around the wall panel edges.

Option	Description	
Wall panel edge reinforcement type		Creates normal reinforcement.

Option	Description	
		Creates U reinforcement.
		No reinforcement is created.
		Creates stirrups.
Mesh	Select whether a mesh is created for the whole panel. If a mesh is created, you can define the mesh properties, or use custom settings. You can use the **Mesh Bars** component to create the mesh. This option is available in **Wall panel reinforcement**.	
Create edge reinforcement	Select whether reinforcement is created at the edges. If you select **Yes**, define the horizontal and vertical reinforcing bar properties. This option is available in **Double wall edge and opening reinforcement**.	

Option	Description
Horizontal reinforcing bars	Define the horizontal reinforcing bar properties for the wall panel edges. Note that horizontal bars have three sets of properties, depending on the length of the bar. For example, if the length of the main bar is less than 3000, it uses the options on the left, if in between 3000 and 6000, the middle options and if greater than 6000, the options on the right. 
Vertical reinforcing bars	Define the vertical reinforcing bar properties for the wall panel edges. Note that vertical bars have two sets of properties, depending on the length of the bar.
U Reinforcement	Select whether to create U rebars and stirrups as a rebar group, or as bent meshes. The bent meshes are always created in the outer layer of the reinforcement. Define the U reinforcing bar properties. Each side of the wall panel has a separate set of U reinforcing bar properties.

Opening tab

Use the **Opening** tab to create the reinforcing bars around the wall panel window and door openings.

Option	Description	
Opening reinforcement type	Can be set for all four opening edges.	
		Creates normal reinforcement.

Option	Description	
		Creates U reinforcement.
		No reinforcement is created.
		Creates stirrups.
Horizontal reinforcing bars	Define the horizontal reinforcing bar properties. Note that horizontal bars have separate sets of properties for the top and the bottom bars.	
Vertical reinforcing bars	Define the vertical reinforcing bar properties.	
U Reinforcement	Select whether to create U rebars and stirrups as a rebar group, or as bent meshes. The bent meshes are always created in the outer layer of the reinforcement. Define the U reinforcing bar properties. Each side of the opening has a separate set of U reinforcing bar properties.	

Door tab

Use the **Door** tab to define the door reinforcement.

Option	Description	
Door reinforcement type	Can be set for all three edges.	
		Creates normal reinforcement.

Option	Description	
		Creates U reinforcement.
		No reinforcement is created.
		Creates stirrups.
Horizontal reinforcing bars	Define the horizontal reinforcing bar properties.	
Vertical reinforcing bars	Define the vertical reinforcing bar properties.	
U Reinforcement	Select whether to create U rebars and stirrups as a rebar group, or as bent meshes. The bent meshes are always created in the outer layer of the reinforcement. Define the U reinforcing bar properties. Each side of the door has a separate set of U reinforcing bar properties.	

Notch tab

Use the **Notch** tab to define the notch reinforcement.

Option	Description	
Notch reinforcement type	Can be set for two edges.	
		Creates normal reinforcement.

Option	Description	
		Creates U reinforcement.
		No reinforcement is created.
		Creates stirrups.
Horizontal reinforcing bars	Define the horizontal reinforcing bar properties.	
Vertical reinforcing bars	Define the vertical reinforcing bar properties.	
U Reinforcement	Select whether to create U rebars and stirrups as a rebar group, or as bent meshes.The bent meshes are always created in the outer layer of the reinforcement. Define the U reinforcing bar properties. Each side of the notch has a separate set of U reinforcing bar properties.	

Diagonals tab

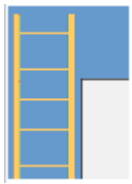
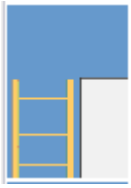
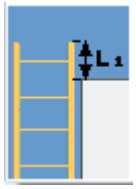
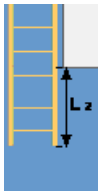
Use the **Diagonals** tab to create diagonal reinforcing bars at the corners of the openings.

Option	Description
Create diagonals	Select whether diagonal reinforcing bars are created or not.
Diagonals count	Number of diagonal reinforcing bars.
Size	Select the diameter of a bar.
Grade	Define the strength of the steel used in the bars.

Option	Description
Bending radius	Define the bar bending radius.
L1 and L2	L1 + L2: Length of the diagonal reinforcing bars.
C	Distance between the diagonal reinforcing bar and the corner of the opening.

Column tab

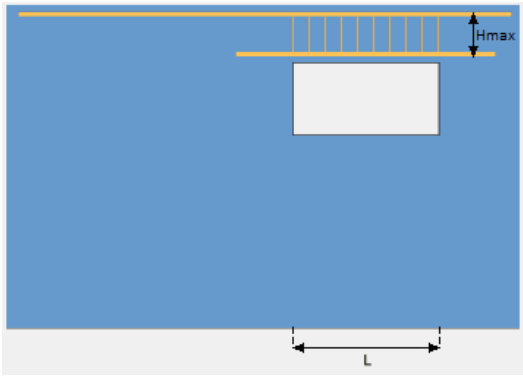
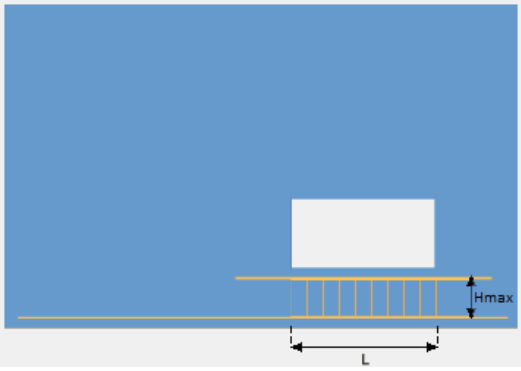
Use the **Column** tab to a create column reinforcement.

Option	Description	
Column reinforcement		Column reinforcement is as high as the part.
		Column reinforcement does not reach higher than the top edge of opening.
		Column reinforcement is approximately the **L** value higher than opening edge.
		Column reinforcement is approximately the **L** value lower than the opening edge.
Column reinforcement properties	Select whether column reinforcing bars are created or not. • **Minimum column height H**: cuts and openings smaller than the minimum value are not taken into account in the column creation. • **Maximum column width B**: if the width between two openings or cuts or edges is greater than the maximum value, then the column reinforcement is not created. • **Column height offset L1**: distance between the column reinforcement end and the opening edge. • **Column height offset L2**: same as L1 but for the bottom part of column reinforcement.	

Option	Description
Main reinforcing bars	Define the main reinforcing bar properties. Note that main bars have three sets of properties, depending on the width of the column.
Stirrup type	Select the stirrup type.
Reinforcing stirrups	Select whether to create rebar stirrups as a rebar group, or as bent meshes. The bent meshes are always created in the outer layer of the reinforcement. Define the stirrup properties. Note that stirrups have three sets of properties, depending on the width of the column.

Beam window top / Beam window bottom tab

Use the **Beam window top** and **Beam window bottom** tabs to create a beam reinforcement on top of the opening and below the opening.

Beam window top	Beam window bottom
	

Option	Description
Beam reinforcement properties	Select whether beam reinforcing bars are created or not. • **Maximum beam height Hmax**: distance from the edge of the wall to the edge of the opening. If the height is greater than the maximum, then the beam reinforcement is not created. • **Minimum beam length Lmin**: distance between the sides of the opening. If the width of the opening is less than the minimum, then the beam reinforcement is not created.

Option	Description
Top	Define the beam top reinforcement properties. Note that the reinforcing bars have two sets of properties, depending on the length of the beam.
Bottom	Define the beam bottom reinforcement properties. Note that the reinforcing bars have two sets of properties, depending on the length of the beam.
Stirrup type	Select whether to create stirrups as a rebar group, or as bent meshes. The bent meshes are always created in the outer layer of the reinforcement. Select the stirrup type.
Reinforcing stirrups	Define the stirrup properties. Note that stirrups have two sets of properties, depending on the length of the beam.

Additional tab
Use the **Additional** tab to create additional horizontal and vertical reinforcing bars.

Option	Description
Additional reinforcing bars	Select whether additional reinforcing bars are created or not. Define the additional reinforcing bar properties.

Attributes tab
Use the **Attributes** tab to control the reinforcing bar properties of the created parts.

Option	Description
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.
Name	Define a name for the part. Tekla Structures uses the name in drawings and reports.
Class	Define the part class number.

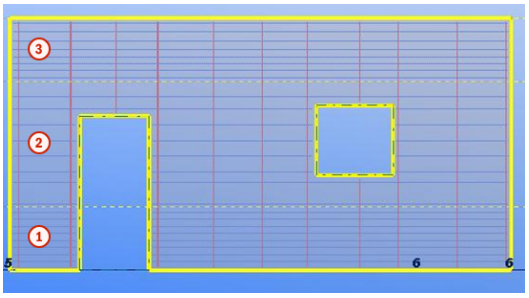
Multiple Wire Size Mesh
Multiple Wire Size Mesh creates reinforcement meshes with multiple reinforcing bar sizes. The created reinforcement mesh is optimized for a mesh welding machine. **Multiple Wire Size Mesh** can be used for slabs and wall

panels. The slabs and wall panels can be of any polygonal shape with any number of openings. The reinforcing bars in the mesh can be of different size, and the spacings between the bars can vary.

Objects created

- Reinforcement meshes

Use for

Situation	Description
	Three zones of longitudinal reinforcing bars

Before you start

Create a polygon slab or a wall panel with any number of openings. Enter zone values to define the width of the area to be reinforced in both the longitudinal and crossing directions. Each direction can be divided up to five zones.

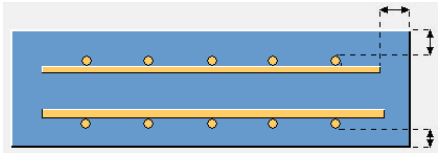
Selection order

1. Select the main part (slab or wall panel).

   The reinforcement mesh is created automatically.

Mesh Parameters tab

Use the **Mesh Parameters** tab to control the mesh creation and cover thickness.

Option	Description
	Define the concrete cover thickness.
Create Rebars	Define to which side of the slab or the wall panel the reinforcing bars are created.
Longitudinal Direction	Define whether the x or the y direction is used as the longitudinal direction when the reinforcing bars are created. By default, the x direction is the longitudinal direction.

Longitudinal Wires/Crossing Wires tab

Use the **Longitudinal Wires** or the **Crossing Wires** tab to control the mesh creation in the longitudinal or the crossing direction, and the reinforcement zones.

Option	Description
Wire sizes	Define the sizes of the reinforcing bars used in the pattern. Separate the sizes with a space.
Min overhang	Define the minimum length of the reinforcing bar extension.
Max overhang	Define the maximum length of the reinforcing bar extension.
Min spacing	Define the minimum space between the reinforcing bars. The value must be divisible by the grid size. For example, if the **Min spacing** is `150`, **Max spacing** is `300` and the **Grid size** is `50`, the spacings are `150`, `200`, `250` and `300`.
Max spacing	Define the maximum space between the reinforcing bars. The value must be divisible by the grid size. For example, if the **Min spacing** is `150`, **Max spacing** is `300` and the **Grid size** is `50`, the spacings are `150`, `200`, `250` and `300`
Grid size	Define the mesh grid size. For example, if the **Min spacing** is `150`, **Max spacing** is `300` and the **Grid size** is `50`, the spacings are `150`, `200`, `250` and `300`. This value depends on the welding machine.
Min wire length	Define the minimum reinforcing bar length.
Min number of welding points	Define the minimum number of crossing reinforcing bar intersections.
Pattern width to fulfill	Define the width of the pattern that is filled with the mesh. This value is used when the zone width is greater than the given value.
Zone width value type	Define the width of the area to be reinforced as a percentage of the whole slab or wall panel width, or as the actual length.

Option	Description
Width, **Real width**, **Mesh area per length**	Define the mesh width, real width and area per length in each zone.
Load pattern from file	Select the external text file where you have defined the patterns for longitudinal and crossing reinforcing bars. If you use the external patterns file, **Multiple Wire Size Mesh** tries to determine a pattern with a suitable area per length value for every zone and apply these values to the zones.

Attributes tab
Use the **Attributes** tab to control the reinforcement properties in the longitudinal and in the crossing directions.

Option	Description
Prefix	Define a prefix for the part position number.
Start Number	Define a starting number for the part position number.
Grade	Define the strength of the steel used in the reinforcing bars.
Name	Define a name for the longitudinal and the crossing reinforcing bars. Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group the longitudinal and the crossing reinforcing bars. For example, you can display the longitudinal and the crossing reinforcing bars of different classes in different colors.

Configuration file for reinforcing bar patterns
Use an external text file to define the patterns for longitudinal and crossing reinforcing bars.

Every pattern starts from the **Pattern** line, followed by, for example, the pattern name. Each **Bar Size** and **Spacing** pair is on a separate row. The **Bar Size** and **Spacing** values are separated by a space.

The file has the following format:

```
< Pattern > < Name >
<Bar Size> <Spacing>
<Bar Size> <Spacing>
```

.....

```
<Bar Size> <Spacing>
```

Examples

Pattern 1

```
10 200
10 200
8 100
8 300
10 400
```

Pattern 2

```
10 200
10 200
8 100
8 100
```

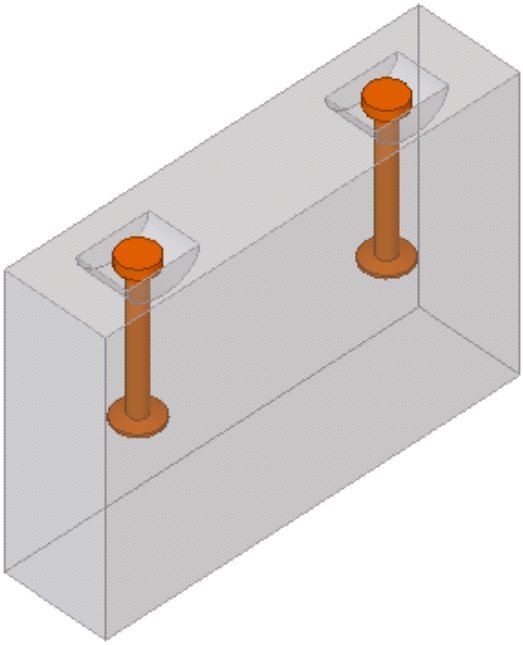
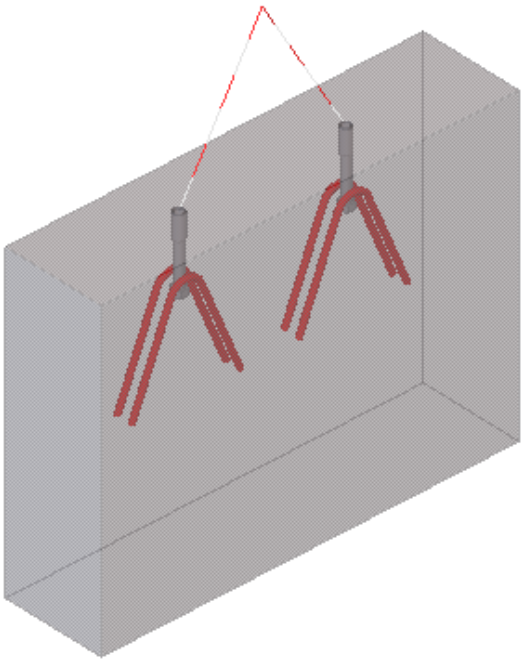
Embedded anchors (8)

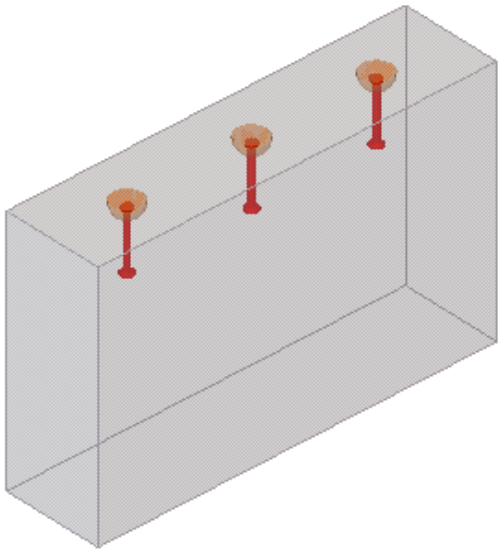
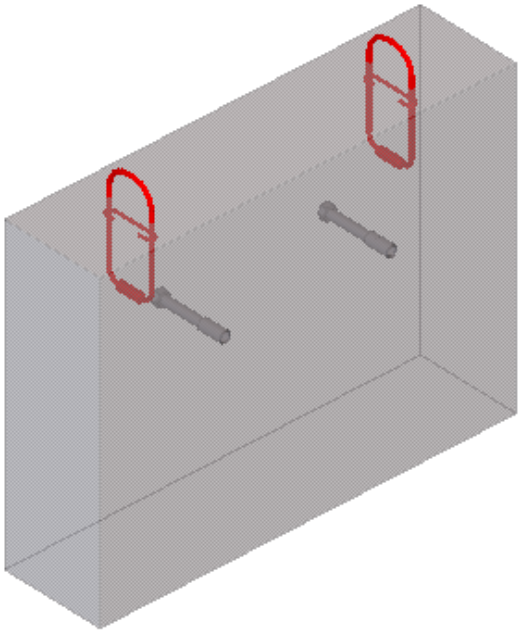
Embedded anchors (8) creates one or more embeds in a concrete part.

Objects created

- Embeds
- Thickening parts
- Reinforcing bars
- Hangup bars
- Crane cables

Use for

Situation	Description
	Two custom anchors.
	Bolt anchors that are custom parts, hangup bars and crane cables.

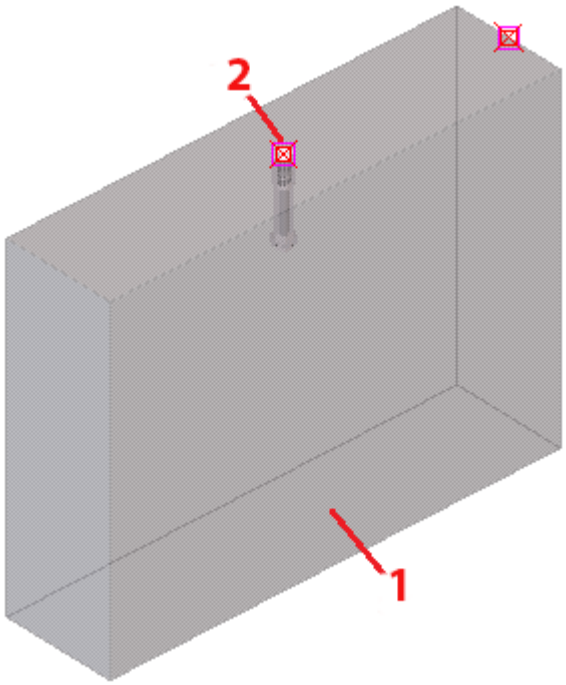
Situation	Description
	Cast-in lifting anchors that are custom parts.
	Cast-in wire rope lifting hoops that are custom parts. Adjusting anchors are in the front side. The component is applied two times.

Selection order

1. Select a concrete part.
2. Pick one point to position the embed.

   You can also select multiple points, depending on how the embeds are divided.
3. Click the middle mouse button to create the embeds.

Part identification key



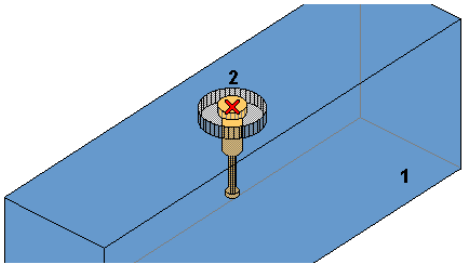
	Part
1	Concrete part (wall, column, beam, slab)
2	Point You can select multiple points.

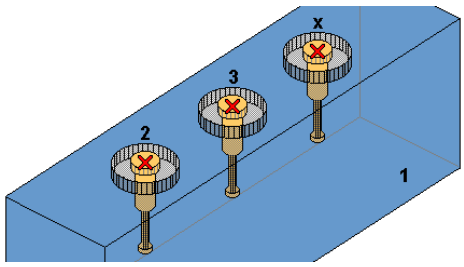
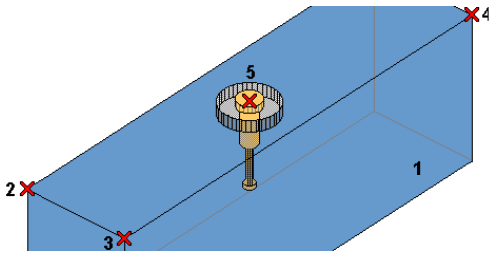
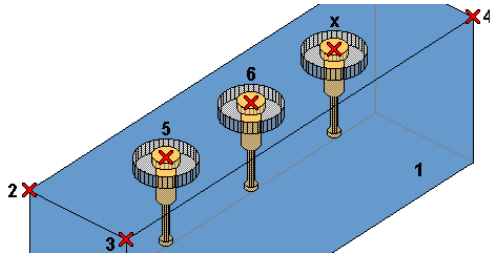
Input tab

Use the **Input** tab to control the custom parts used for the embeds, the input points for placing the embeds and the component direction properties.

Input point sequence

Select the distribution method for the embeds. You can also adjust the placement of embeds on the **Input** tab.

Option	Description
	Select a concrete part and pick one position point for the anchor.

Option	Description
	Select a concrete part and pick multiple position points for the anchors.
	Select a concrete part and pick three points to define the plane. Pick a point to place the embed.
	Select a concrete part and pick three points to define the plane. Pick multiple points to place the embeds.

Up direction

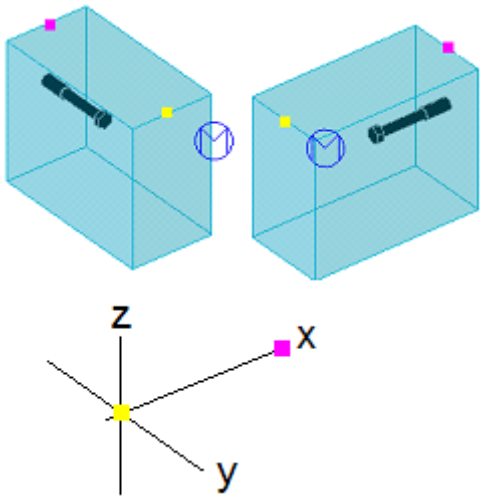
Define how the embeds are rotated.

This option is not available if the three-point-plane option is selected from the **Input points sequence** list.

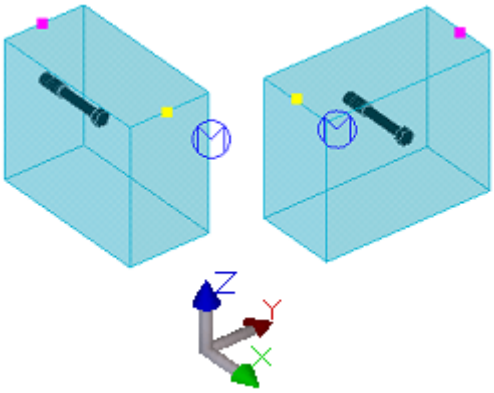
You can use the global directions +X,-X,+Y,-Y,+Z,-Z or the local directions +X,-X,+Y,-Y,+Z,-Z to define a specific face of a typical cube-shaped (slab, wall, beam of column) part.

Alternatively, you can use the **Top in form face** option. This option aligns the embedded anchors with the top form face of the cast unit.

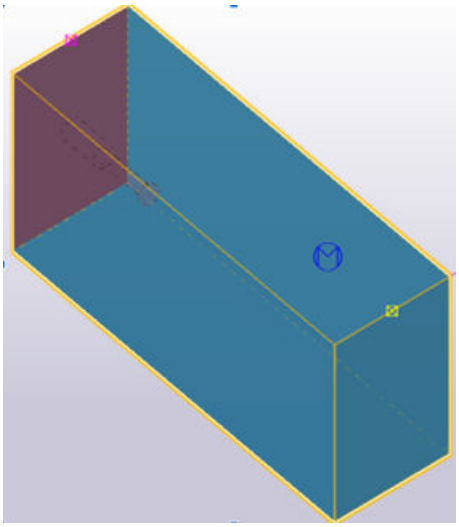
-X local:



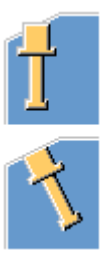
+X global:

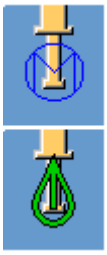


Top in form face:



Embed handling



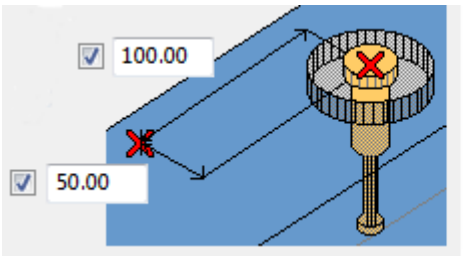


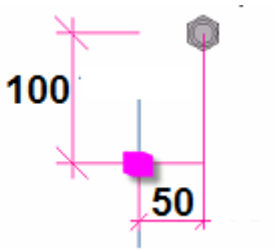
Select whether the embed is handled as a detail or as a modeling tool.

In general, a detail uses the shape of the main part for positioning the embeds, and a modeling tool uses the picked points for positioning the embeds.

Embed offset

Define embed offsets in X- and/or Y-direction, from the picked position points.





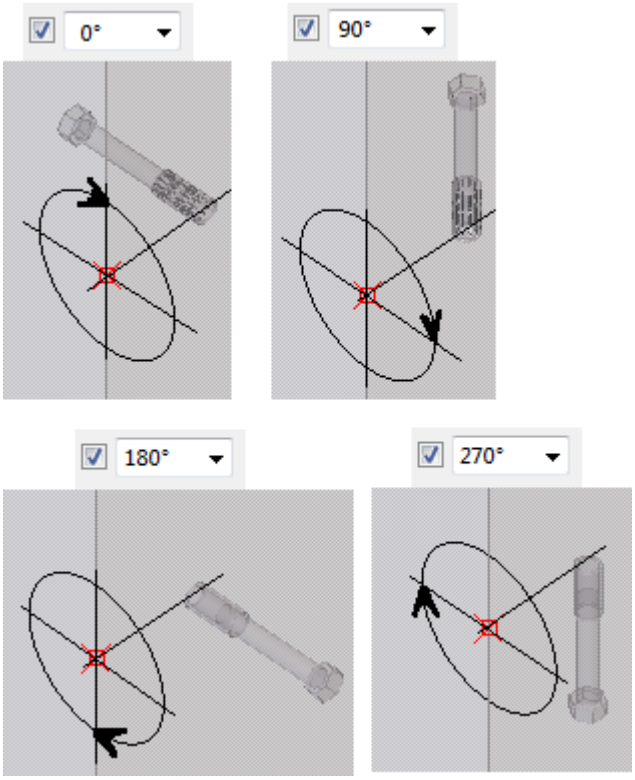
Anchor type

Select the type of the anchors from the **Anchor type** dropdown list. The options are:

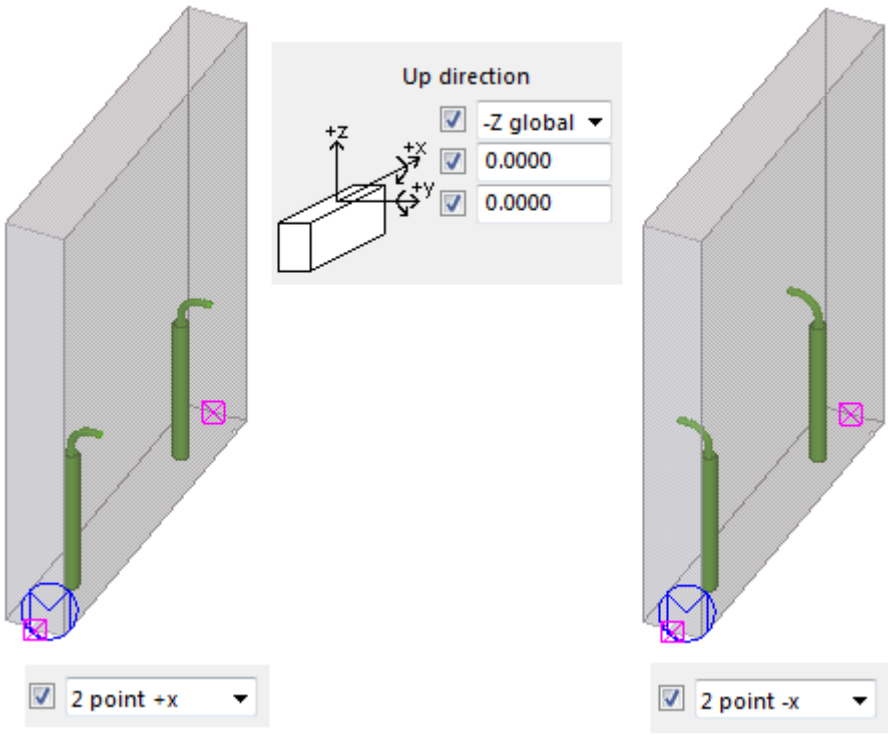
- **Anchor created by top part and bottom part**
- **Component part**
- **Component detail**
- **Surface treatment**
- **Component part and component detail**
- **Component part and top and bottom part**
- **Component detail and top and bottom part**

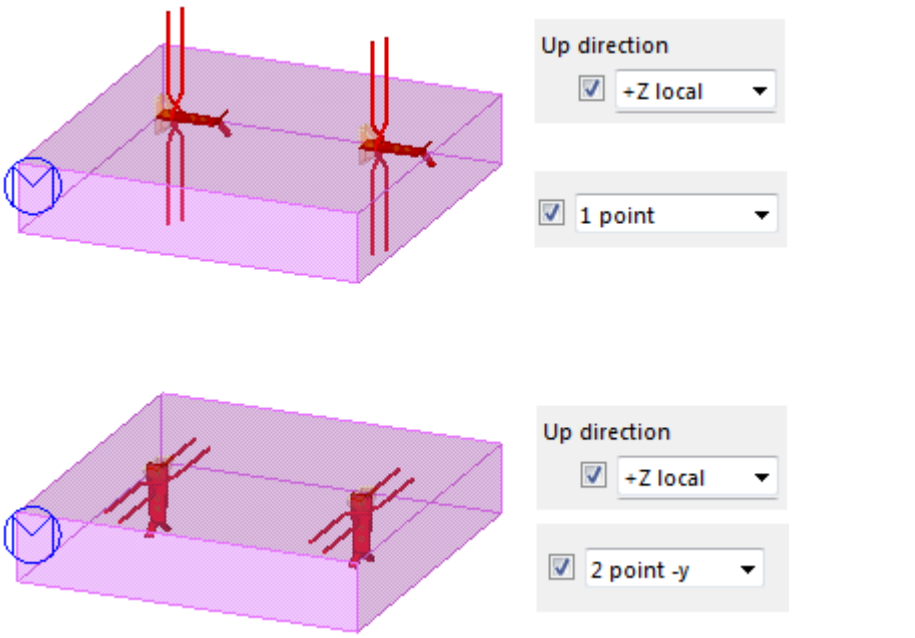
Custom part

Option	Description
Partname component	1. Select a custom part from the **Applications & components** catalog to be used as embed. 2. Select the configuration file.

Option	Description
	3. Select whether the custom parts must be used for the embeds. • If set to **Do not connect**, define the embeds' shape on the **Input** tab. • Use the other options to connect the custom part to the main part as welded, added material, add to cast unit, or add as sub-assembly. 4. Select how the embeds are rotated. The default direction is **2 point +x**. 5. Select how the embeds are rotated around a position point.  6. Check the top in form face in direction to adapt the direction of the custom part.
Detailname component	1. Select a custom detail from the **Applications & components** catalog to be used as embed. 2. Select the configuration file.
Surface treatment	1. Define the thickness, width, and height of the surface treatment. 2. Enter the name, color, and subtype.

Custom part positioning examples





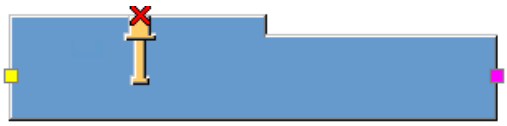
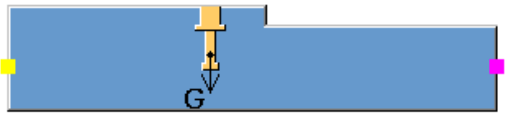
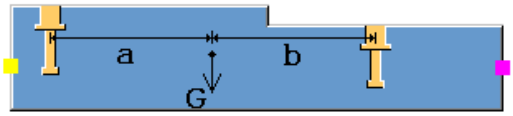
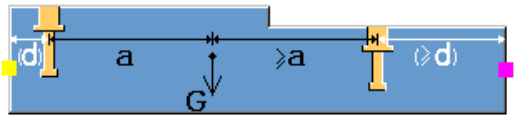
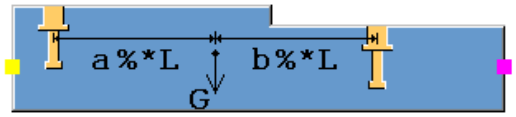
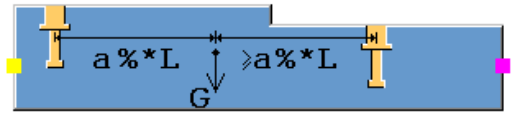
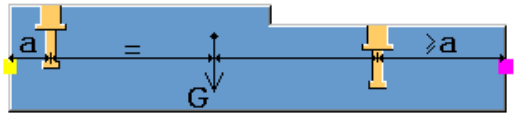
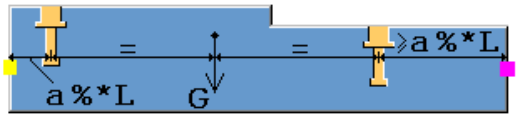
Placement tab
Use the **Placement** tab to control the placing of the embeds, crane cable angles and safety factor properties and to select the lifting anchors.

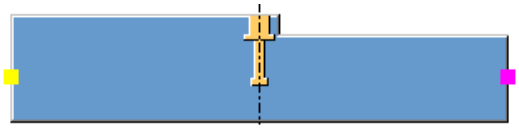
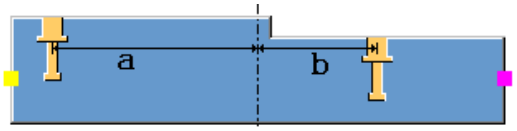
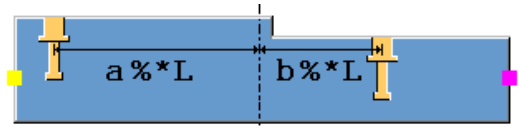
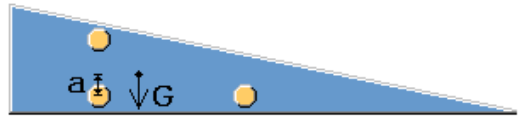
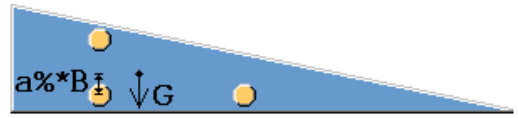
You can define the input point sequence, lift system and embed dimensions both in reference line and cross section direction.

Input points sequence

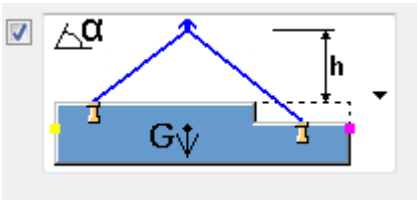
Select the distribution method of the embeds.

Use the **a** and **b** boxes on the right to enter the values.

Option	Description
	Embed is placed on a selected position point.
	Embed is placed on a center of gravity (COG) point in the part length direction.
	Multiple embeds. Define dimensions **a** and **b**. Reference = COG
	Define dimensions **a** and **b**. Reference = COG If embeds are likely to be placed outside the part, then value **d** is used.
	Define dimensions **a** and **b** as percentages of the part length. Reference = COG
	Define dimension **a** as a percentage of the part length. Reference = COG
	Define dimension **a**. The right embed is calculated from COG.
	Define dimension **a** as a percentage of the total part length. Reference = COG

Option	Description
	Embed is placed in the middle of the part.
	Define dimensions **a** and **b**. Reference = middle of the part
	Define dimensions **a** and **b** as percentages of the part length. Reference = middle of the part
	Define dimensions **a** and **b**. The distances are from the embeds to the part ends.
	Define dimensions **a** and **b** as percentages of the part length. The distances are from the embeds to the part ends.
	Three embeds in a triangular plate. Define the vertical distance **a** from COG.
	Three embeds in a triangular plate. Define the vertical distance **a** from COG as a percentage of the width **b**.

Lift system



Select the crane cable lifting system.

Dimensions

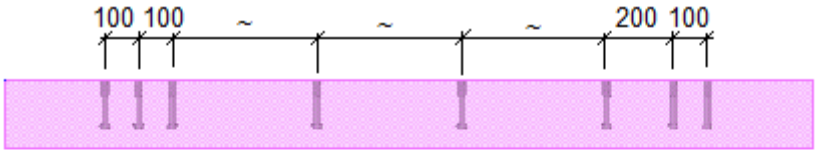
Option	Description
a	Define embed dimension **a**.
b	Define embed dimension **b**.

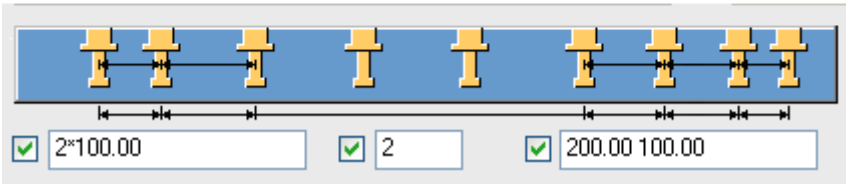
Option	Description
c	Define distance **c**.
d	Define anchor distance from the part edge.
h	Define crane cable height.
α	Define crane cable angle **α**.
β	Define cable angle **β**.
Rounding	Define the rounding value for anchor distances.

Additional embeds

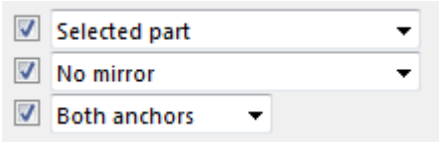
Define additional embeds. Use the boxes on the left and right to define distances. You can enter multiple distances. Use a space to separate the values.

Use the middle box to define the number of additional embeds.





Mirroring



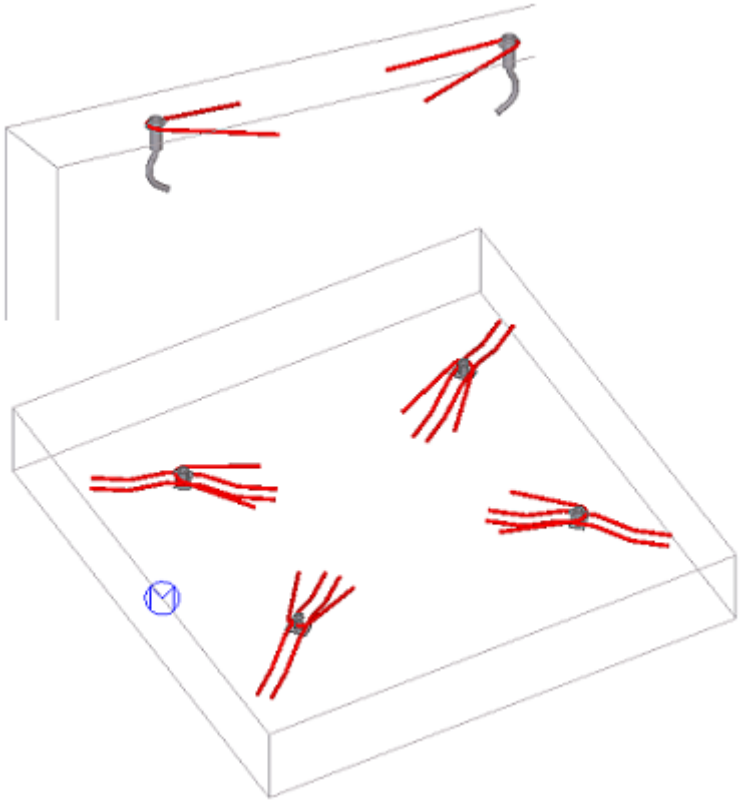
- Define the COG (center of gravity) for the selected parts, all connected parts, all connected parts without steel, an assembly, or assembly without steel parts.

  All connected parts and **All connected parts without steel** include parts connected by bolts and welds.

  Assembly and **Assembly without steel parts** include the parts in the current assembly only.

- Select whether embeds are mirrored.

  Mirroring is useful for asymmetric custom part embeds. You can mirror both in reference line direction and cross section direction.



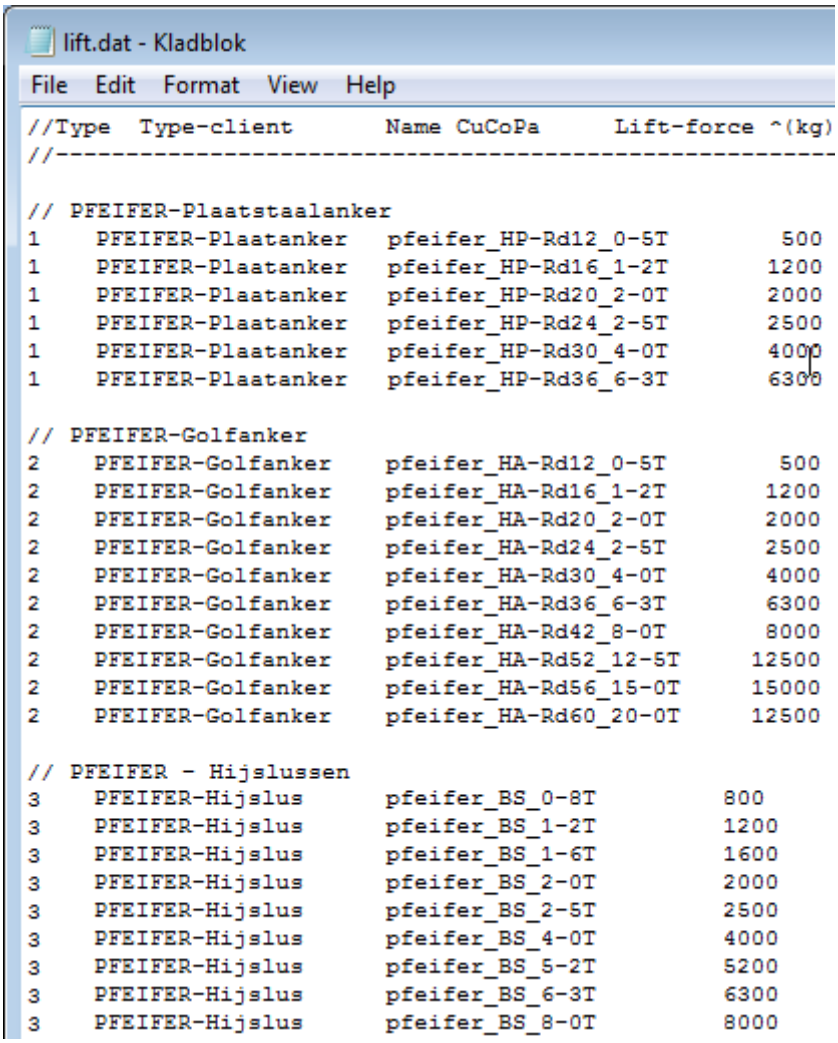
- Select whether both anchors are created, or only left or right anchor.

Lift.dat configuration file

You can use a configuration file to calculate the lifting capacities of lifting anchors.

Set **Browse configuration file** to **Yes**, and define the full path to the configuration file. The default file is `lift.dat`, located in the `profil` folder.

The `lift.dat` configuration file contains a list of all custom lifting anchors belonging to the default Tekla Structures installation. The anchors are sorted based on fabricator and type, component name and lifting capacity. You can add more custom components to the list in the configuration file. The file can be opened in any text editor, for example Notepad.



- **Type**

  Groups where lifting anchors can be selected.
- **Type client**

  Description of types.
- **Name CuCoPa**

  Name of custom component part as listed in the **Applications & components** catalog.
- **Lift force (kg)**

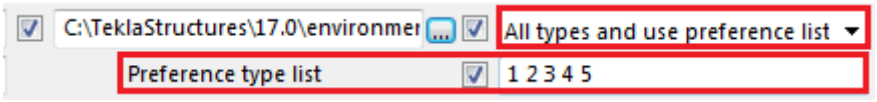
  Lifting capacity in kilograms.
- **Dir**

  Direction. Component direction during creation of custom component part.
- **Attribute file**

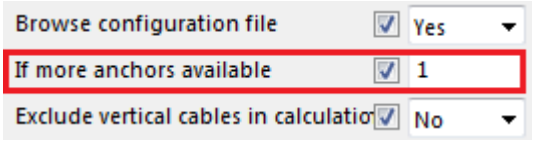
  Link to the configuration file.

The anchor type can be selected from the list. The first list option corresponds to the first type (`1`) in the `lift.dat` configuration file.

The last list option is **All types and use preference list**. This option searches from all groups. A preference can be defined in the **Preference type list**. Start with the highest preference group and separate the groups with a space.



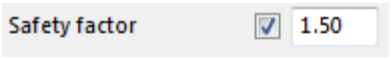
Several fabricators have lifting anchors with the same lifting capacity, but with different anchor lengths. Use the order in the configuration file to define which anchor is selected. First found anchor with correct capacity = 1, second found anchor = 2, and so on.

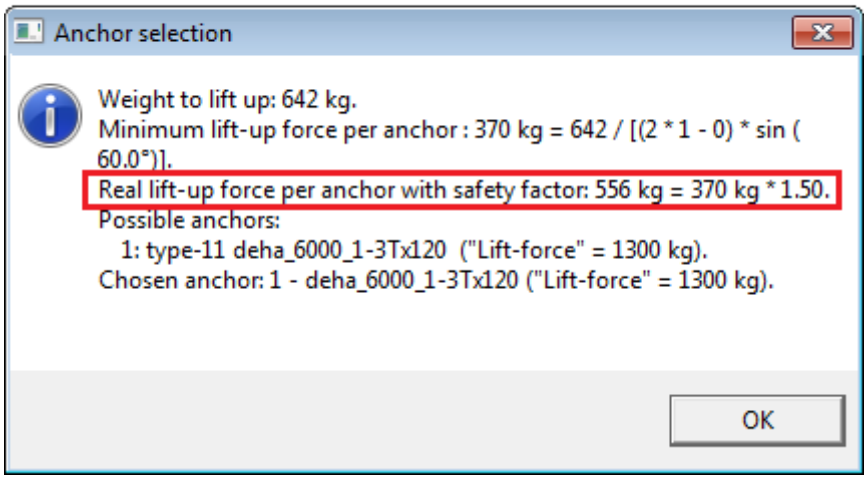


NOTE If you have selected to use a custom part, ensure that the corresponding custom parts are available in the model. Names of the custom parts should not contain spaces, because they are not read from the `lift.dat` configuration file.

Safety factor

Define the safety factor. The weight to be lifted up is multiplied with this factor in the anchor calculation.





Braced girder

Select to move the anchors to the closest braced girder in **Move to closest braced girder**.

Enter the **Maximum displacement** and **Placement offset** of the anchors.

Define the **Class** and **Name** for the top bar in the braced girder. Select whether to detect braced girders in the whole assembly.

Avoid reinforcement

Select whether all reinforcement types are detected in the main part and define the placement offset to avoid collision with anchors.

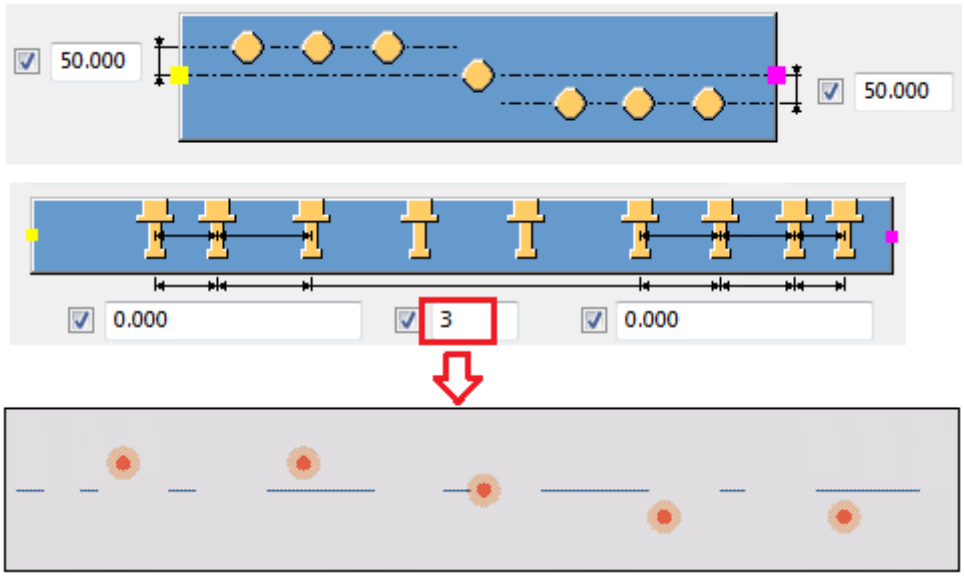
Define the **Class** and **Name** for the reinforcement. Select whether to detect reinforcement in the whole assembly.

Top placement tab

Use the **Top placement** tab to define the embed offsets and how the information message box and crane cables are shown.

Offset

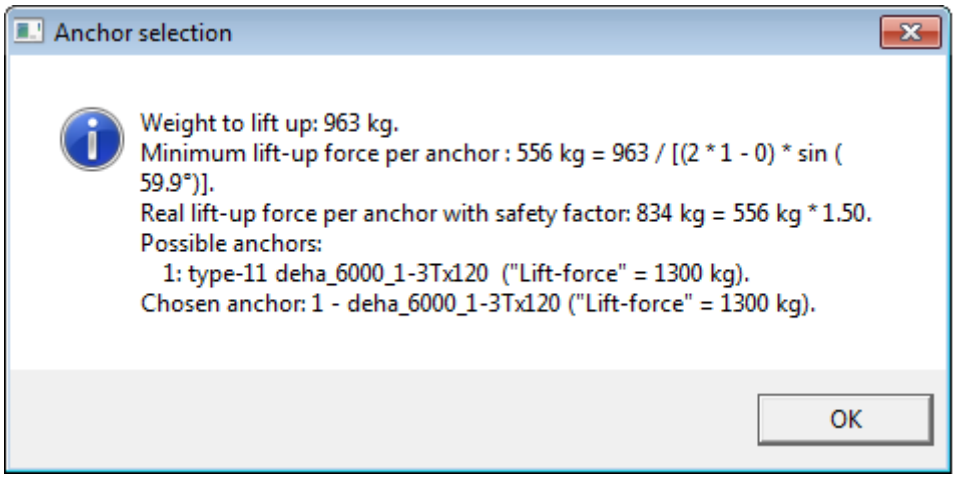
Define the embed offsets at both ends of the concrete part. An embed which is positioned exactly in the middle of the concrete part stays in that position and does not get an offset.



Show information message boxes

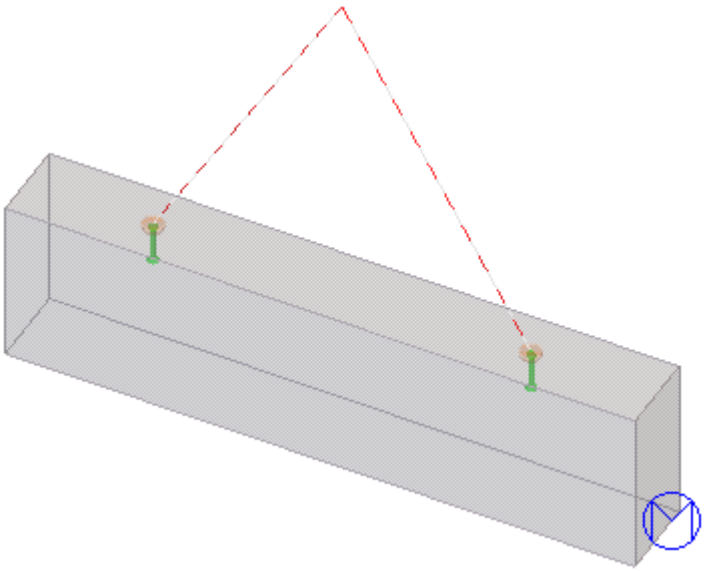
Define whether a message box containing the embed calculation results is shown. The message box shows the weight to be lifted up, the real lift-up weight including a safety factor, and the selected anchor.

The message box appears only if you have set **Browse configuration file** to **Yes** on the **Placement** tab.



Show crane cables

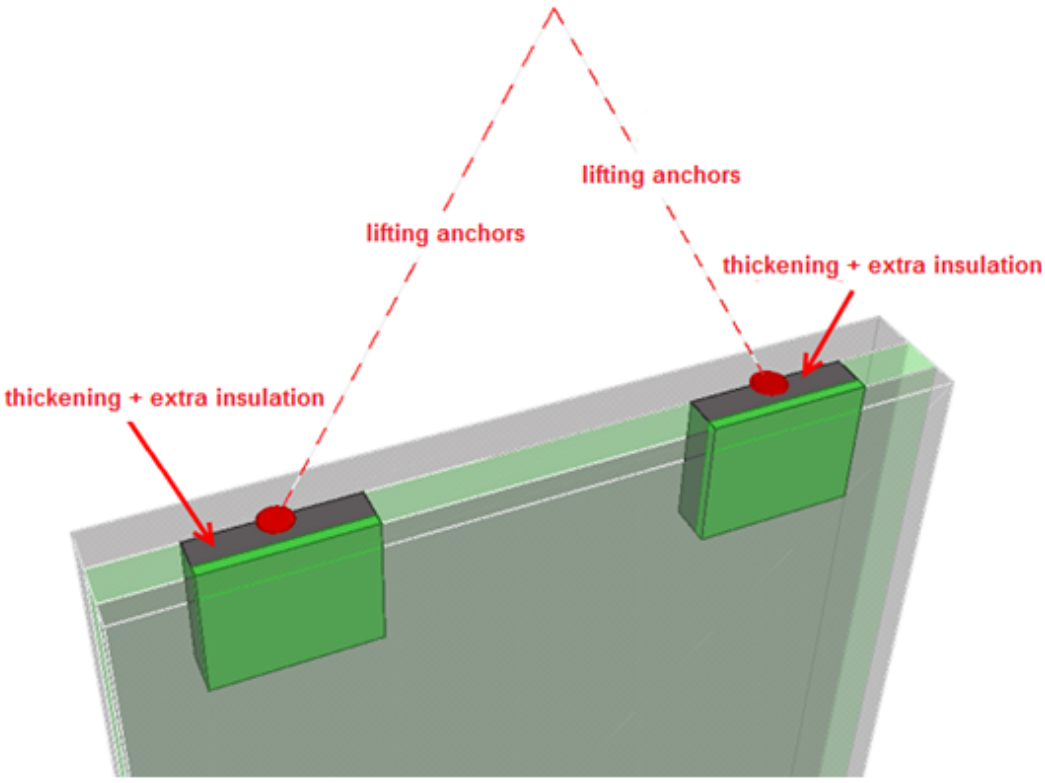
Define whether crane cables are shown. If set to **Yes**, crane cables are shown as construction lines or profiles with the **α** angle, defined on the **Placement** tab.



Thickening tab

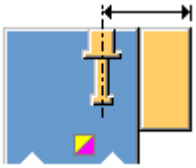
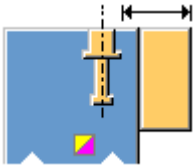
Use the **Thickening** tab to define the thickening of the concrete layer. The thickening is especially useful for sandwich walls where shells are thickened near lifting anchors.

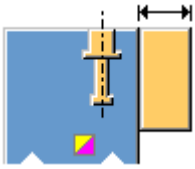
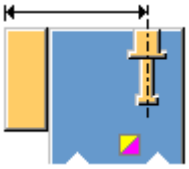
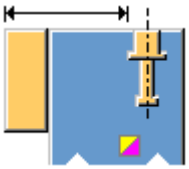
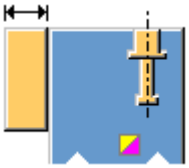
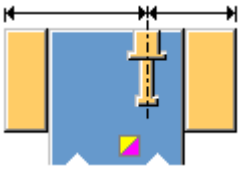
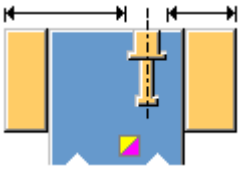
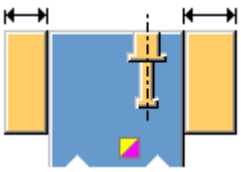
In the example below, a thickening is added with an extra insulation layer.



Thickening options

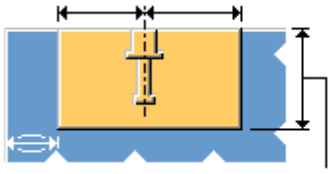
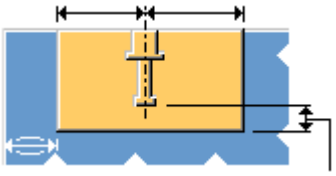
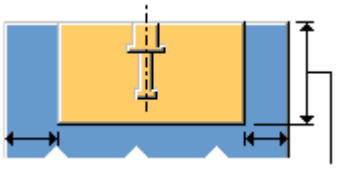
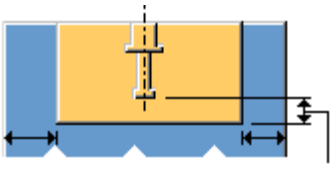
Select the side where thickening is applied. You can also define the thickness.

Option	Description
	No thickening.
	Right side thickening. Thickness is defined from center of the embed.
	Right side thickening. Thickness is defined from side of the embed.

Option	Description
	Right side thickening. Thickness is defined from side of the shell.
	Left side thickening. Thickness is defined from center of the embed.
	Left side thickening. Thickness is defined from side of the embed.
	Left side thickening. Thickness is defined from side of the shell.
	Thickening on both sides. Thickness is defined from center of the embed.
	Thickening on both sides. Thickness is defined from side of the embed.
	Thickening on both sides. Thickness is defined from side of the shell.

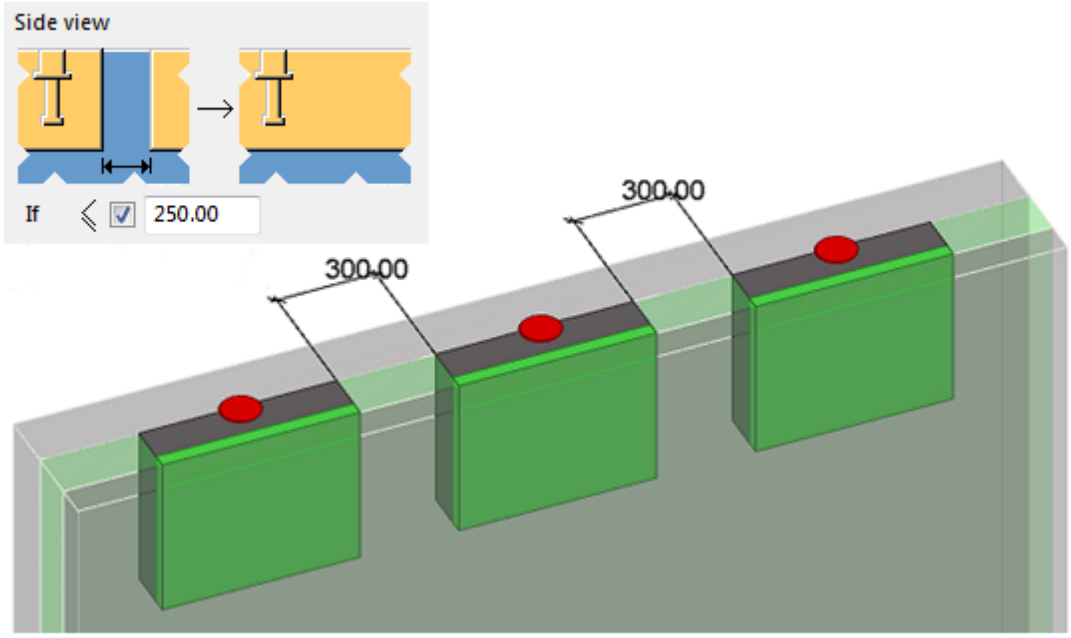
Thickening width and height

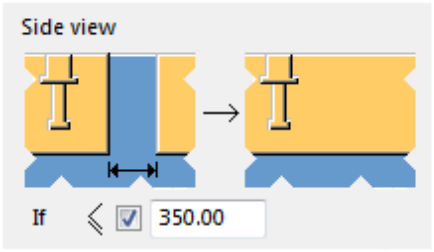
Select the thickening part reference, and define the thickening width and thickening height.

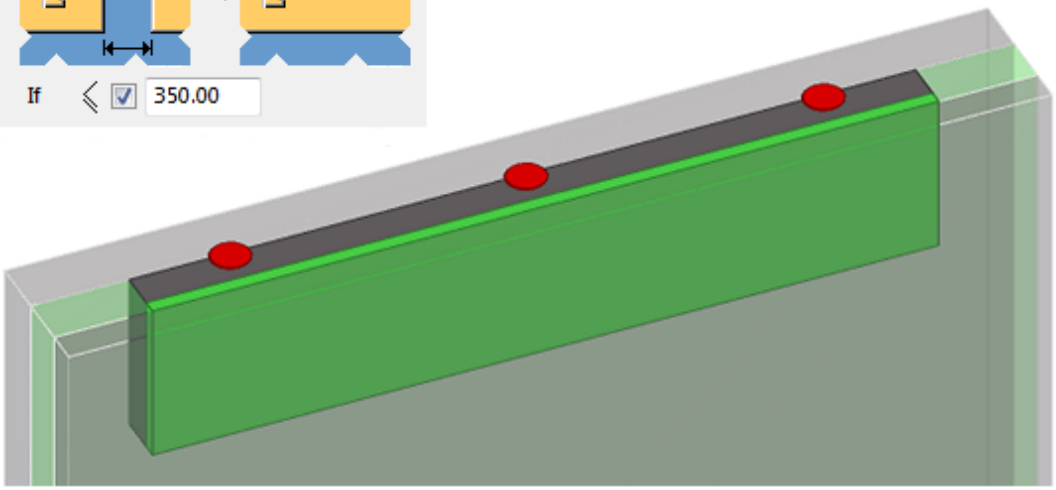
Option	Description
	Width is defined from center of the embed. Height is defined from the top side of concrete.
	Width is defined from center of the embed. Height is defined from the bottom side of concrete shell.
	Width is defined from the side of concrete shell. Height is defined from the top side of concrete shell.
	Width is defined from the side of concrete shell. Height is defined from the bottom side of embed.

Side view

Option to combine multiple thickening parts into one if the distance between the thickenings is smaller than or equal to the defined value.





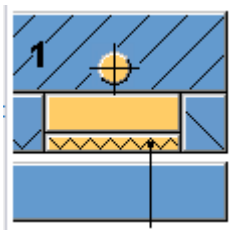


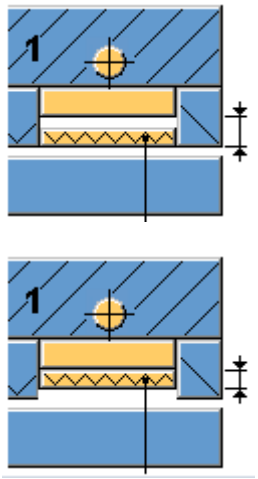
Thickening part

Select how the thickening is connected to the shell. Define the thickening properties.

Additional insulation

Select how to add insulation to thickening part.

Option	Description
Insulation classes	Define the class of the insulation which is used in the concrete shell. If the defined class number matches the class of the insulation in the model, then the insulation will be cut at the location of the thickening.
Foil classes	Define the class of the insulation which is used in the sandwich wall. If the defined class number matches the class of the insulation, then the insulation will be cut at the location of the thickening
Create extra insulation	Select whether an additional insulation layer is created.
	Define the thickness of the insulation.

Option	Description
	

Additional options for the insulation part

Option	Description
Extra insulation part	Select whether the insulation part is added, and how it is added.
To	Select the part to which the insulation part is added.

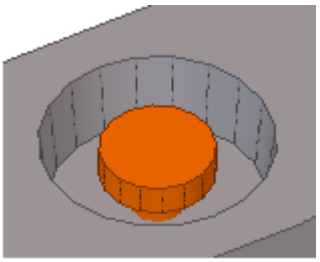
Picture tab

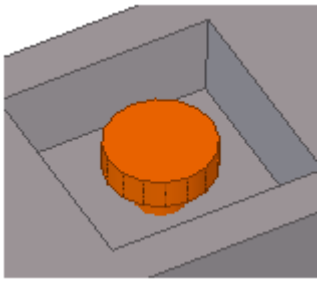
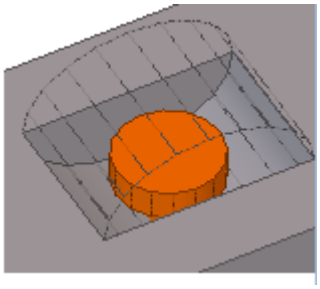
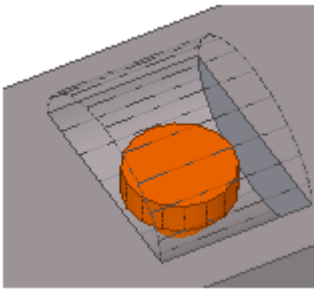
Use the **Picture** tab to control the embed dimensions, connection method and rotation.

Embed dimensions

Define if the embed needs to be recessed. You can define the dimensions of the recess, distance from the recess plane to the top of the embed and select the shape of the cutout.

Select the shape of the cutout.

Option	Example
Circle	

Option	Example
Square	
Half moon X	
Half moon Y	

As negative volume

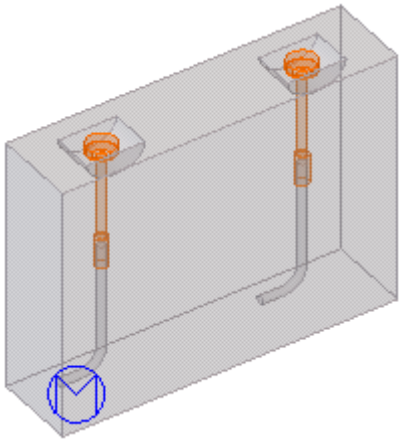
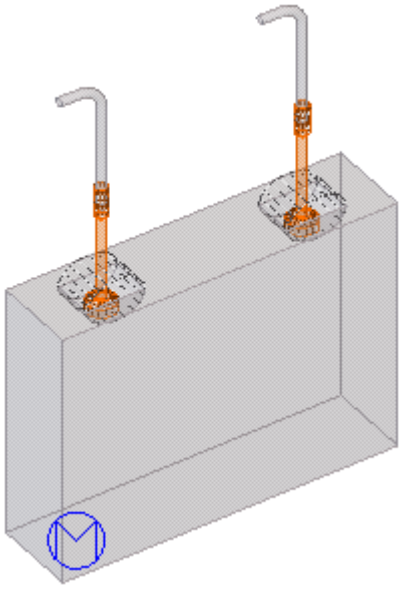
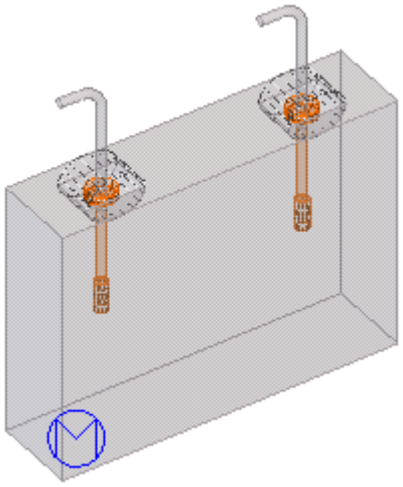
Select whether cutout profiles are handled as empty cutouts or cutouts with a formwork part.

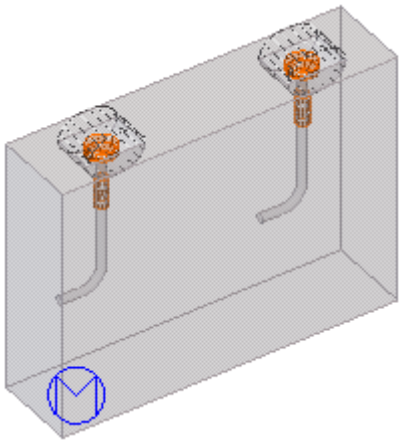
Top part/Bottom part

Set the part class and rotation. Each option rotates the embed 90 degrees counterclockwise. You can define also a fixed rotation angle.

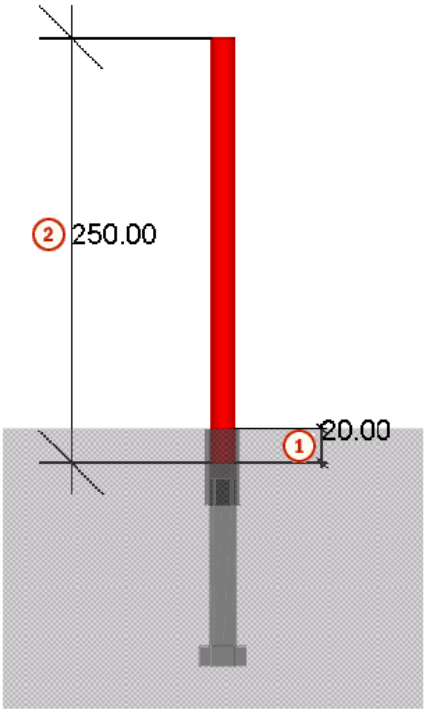
Positioning

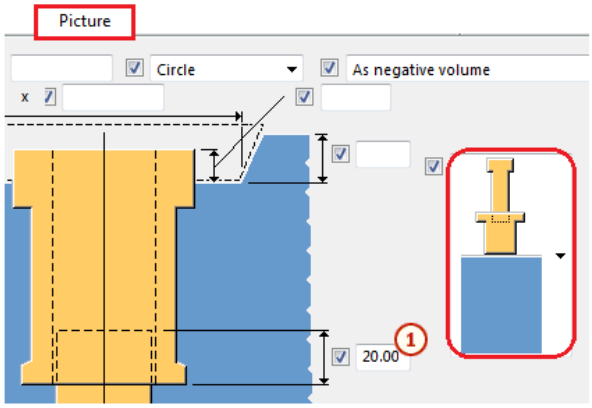
Select how to position the top and the bottom parts of the embed.

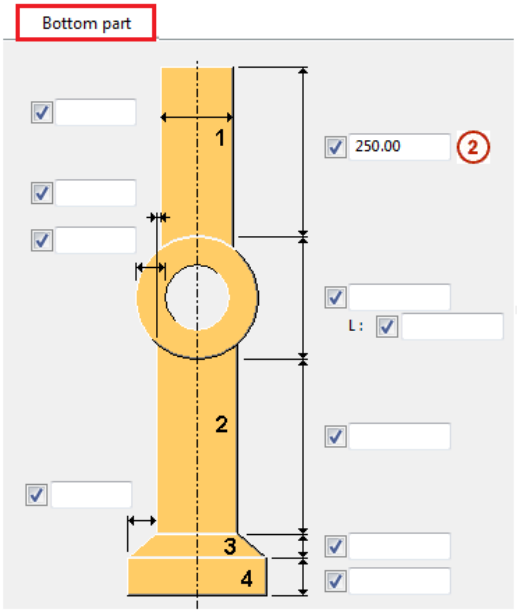
Option	Example
	
	
	

Option	Example
	

An example of an anchor that is positioned on top of the concrete part.







Create top part

Select whether the top part of embed is created, and if set to **Yes**, select how the part is connected to concrete part.

Create bottom part

Select whether the bottom part of embed is created, and if set to **Yes**, select how the part is connected to the concrete part.

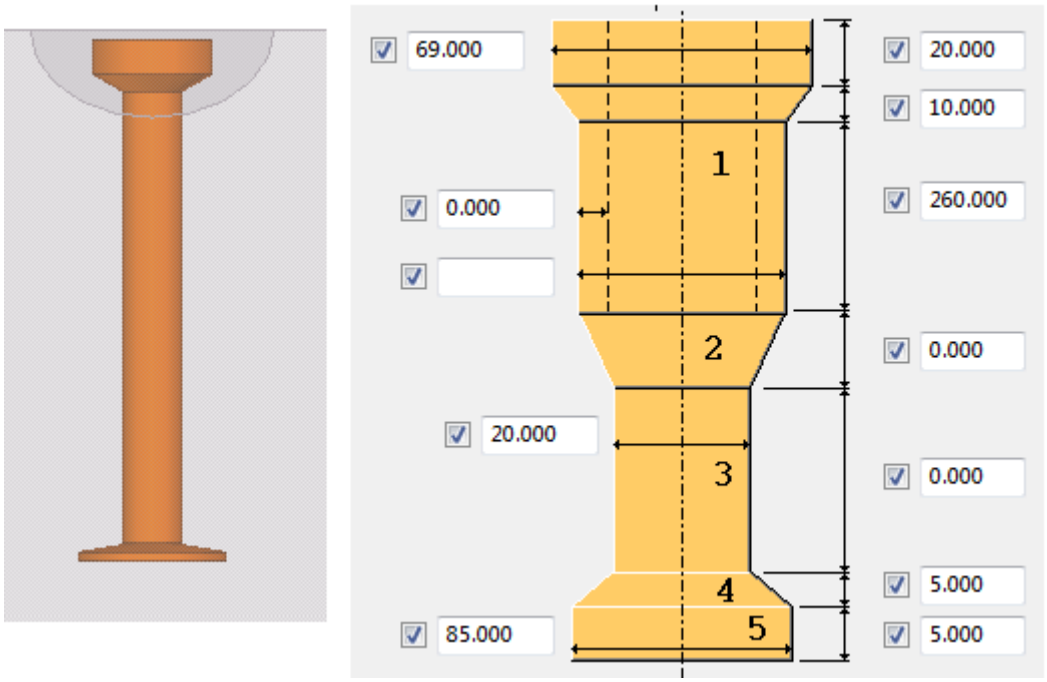
Top part tab

Use the **Top part** tab to define the top part of the embed.

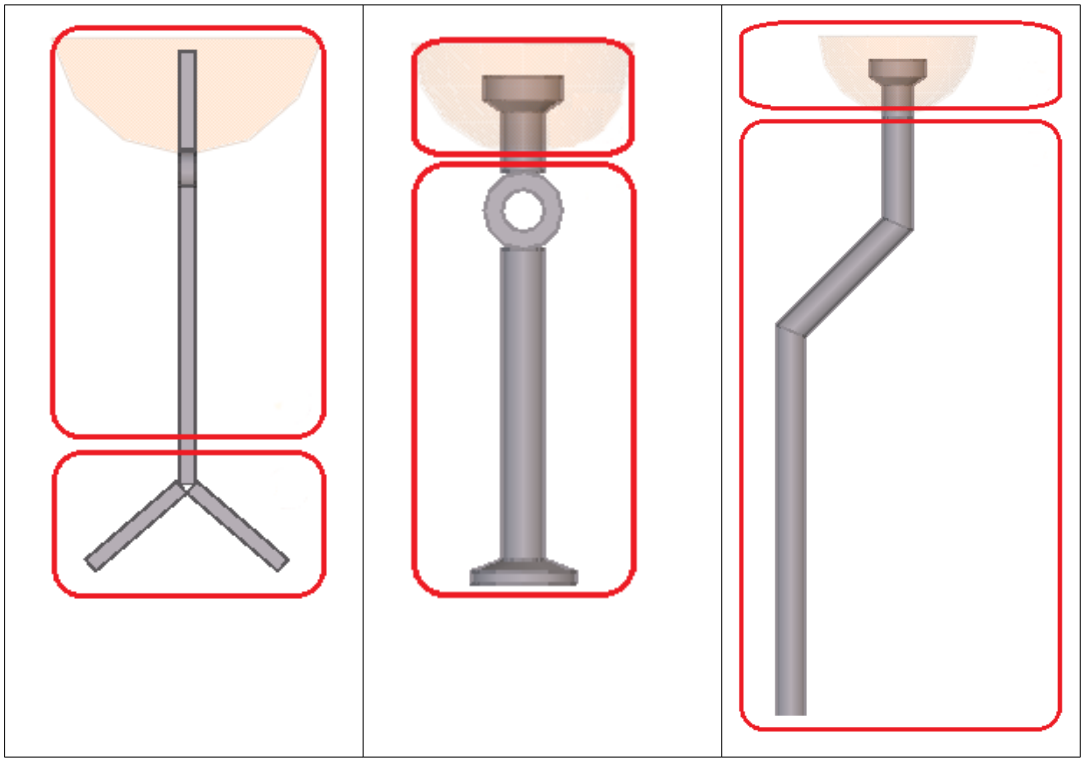
Top part dimensions

Use the diameter and height boxes to define the shape of the top part of the embed. You can define the main diameters of parts 1, 2, 3, 4 and 5 also on the **Parts** tab.

Examples



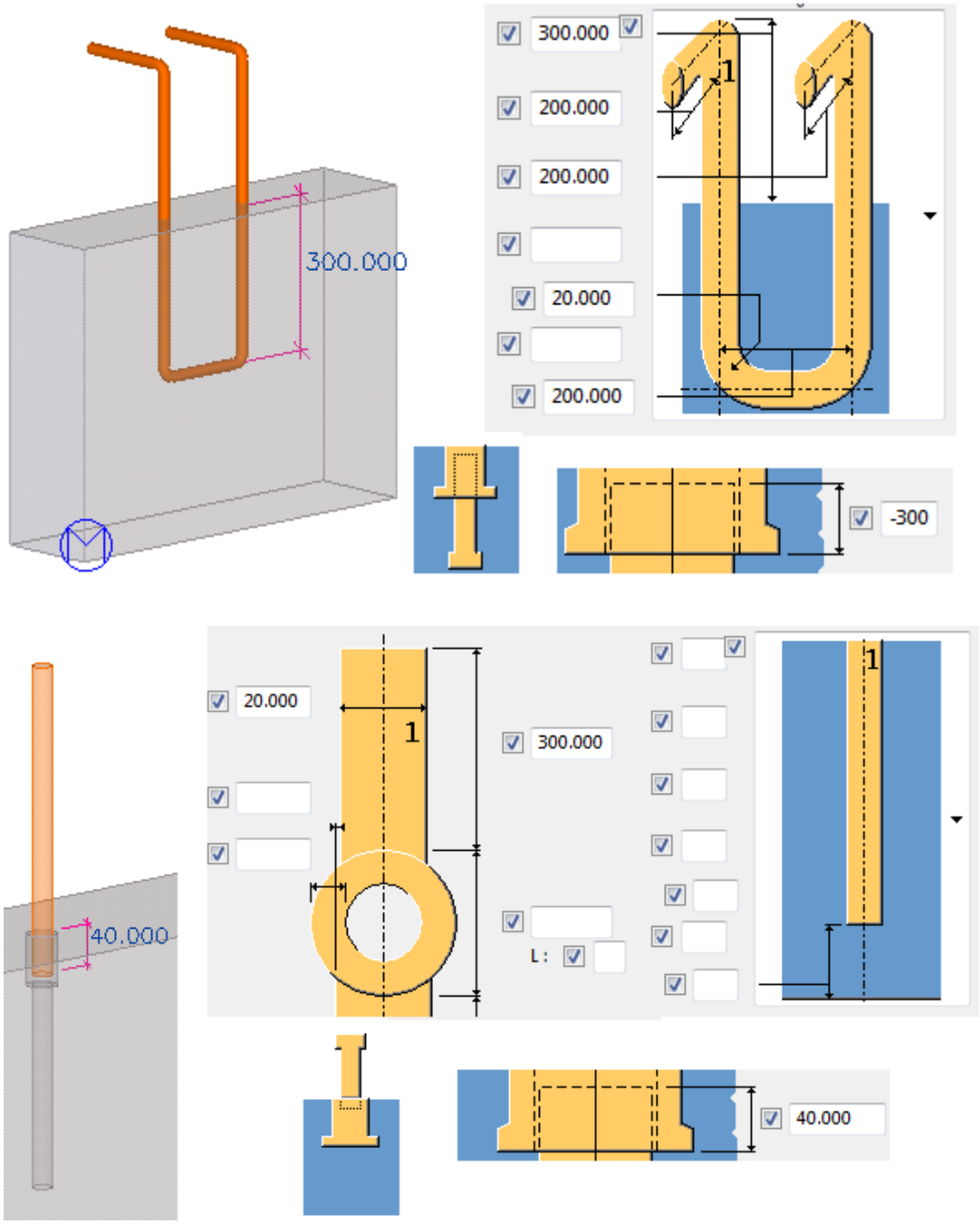
Embed top part defined on the **Top part** tab, embed bottom part defined on the **Bottom part** tab.



Bottom part tab
Use the **Bottom part** tab to define the bottom part of the embed.

Bottom part dimensions

Use the diameter and height boxes to define the shape of the bottom part of the embed. You can define the main diameters of parts 1, 2, 3, 4 and 5 also on the **Parts** tab.



Parts tab

Use the **Parts** tab to define the embed top and bottom part properties and the formwork part properties.

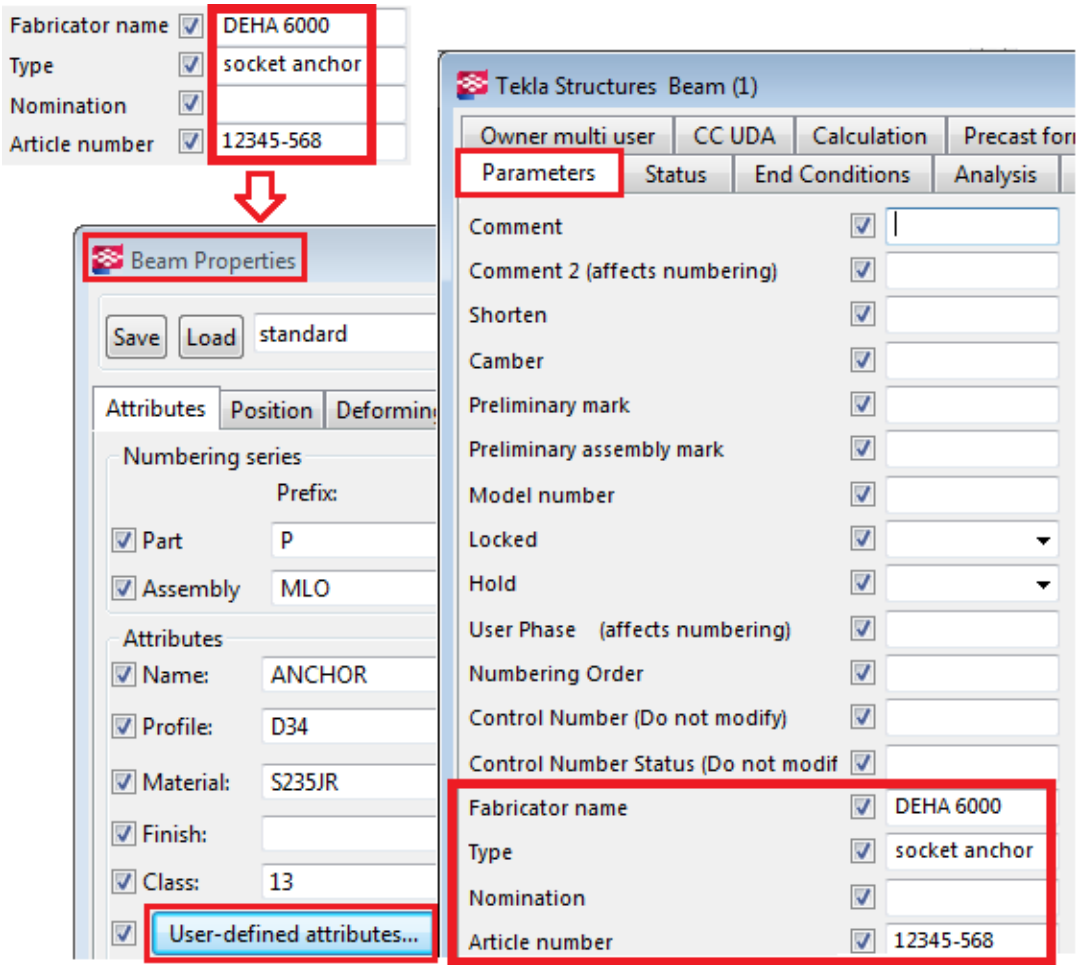
Part properties

Both the top and the bottom part are build from multiple profiles. You can define profiles for each section.

Define the part properties for the top, bottom and formwork part. If the profile properties are left empty, the lengths and diameters defined on the **Top part** and **Bottom part** tabs are used.

Option	Description
t, **b**, **h**	Part thickness, width and height.
Pos_No	Prefix and a start number for the part position number.
Material	Material grade.
Name	Name for the part.
Comment	Add a comment for the part.

You can define UDAs for the top and bottom parts.

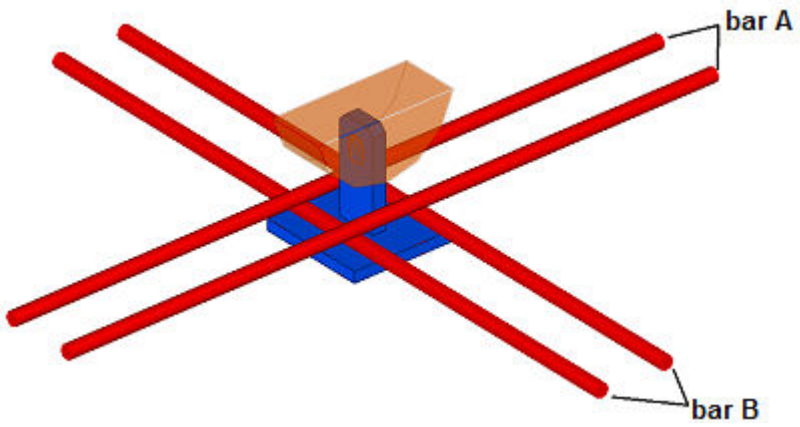


Reinforcing bar tab

Use the **Reinforcing bar** tab to define additional reinforcing bars for the embeds.

You can define the reinforcing bar shape and hook properties, and the reinforcing bar profile properties in two directions.

Example

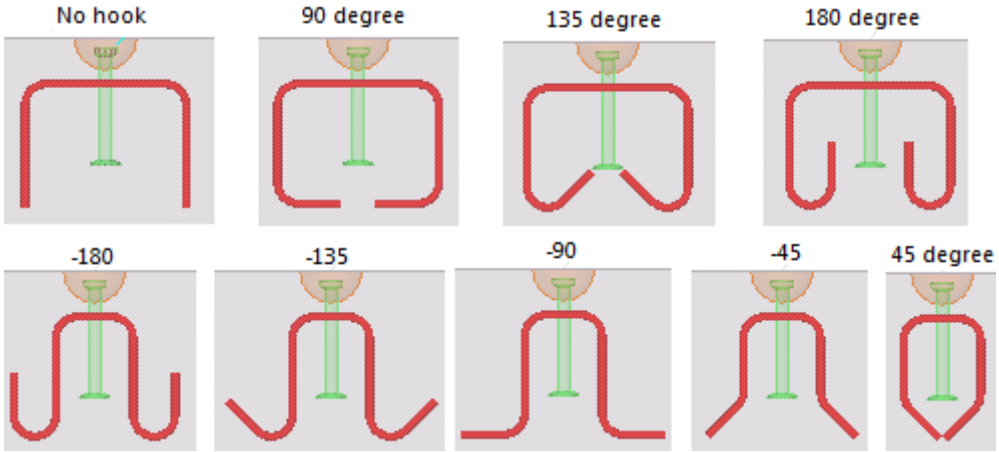


Grade, Size

The **Grade** and **Size** options work together. Click the **...** button next to the **Size** box to open the reinforcing bar catalog and to select a grade and a size for **bar A** and **bar B**.

End conditions left/End conditions right

Select the shape of the reinforcing bar.

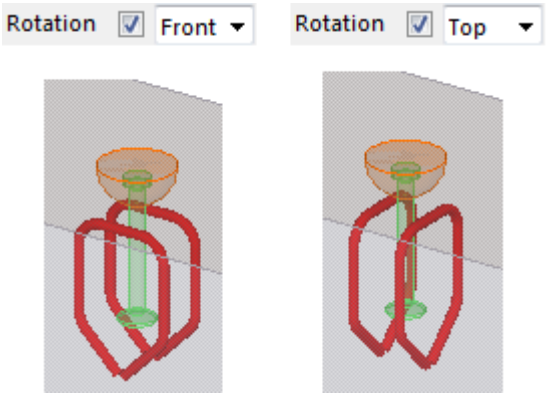


Bend lengths left/Bend lengths right, Bend radius

Define the bend length for the hooks and the bend radius.

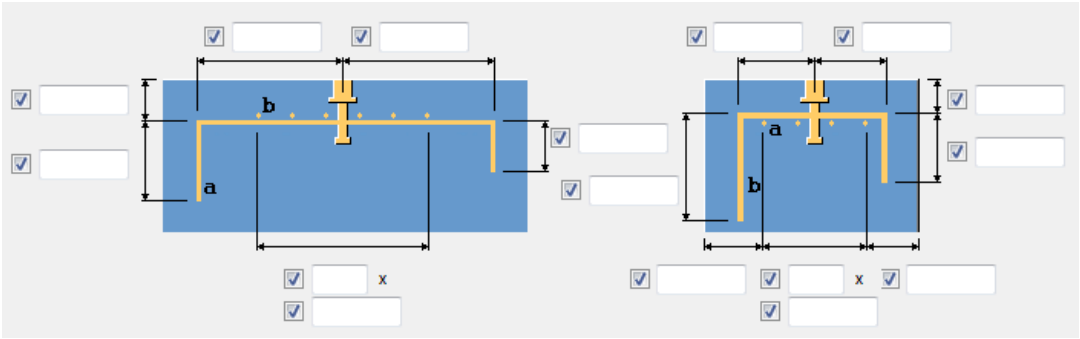
Rotation

Select how to rotate the reinforcing bars and define the rotation angle.



Dimensions

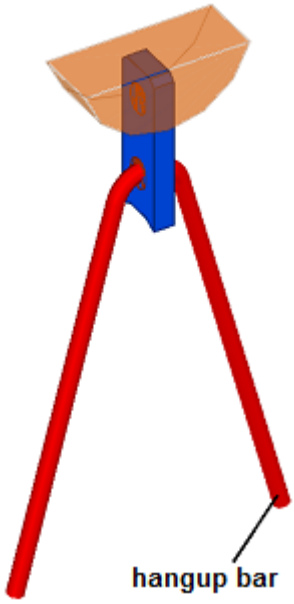
Define the reinforcing bar dimensions and the number of bars.



Hangup bars tab
Use the **Hangup bars** tab to define additional reinforcing bars for the embeds.

You can define the reinforcing bar shape and hook properties, and the reinforcing bar profile properties.

Example

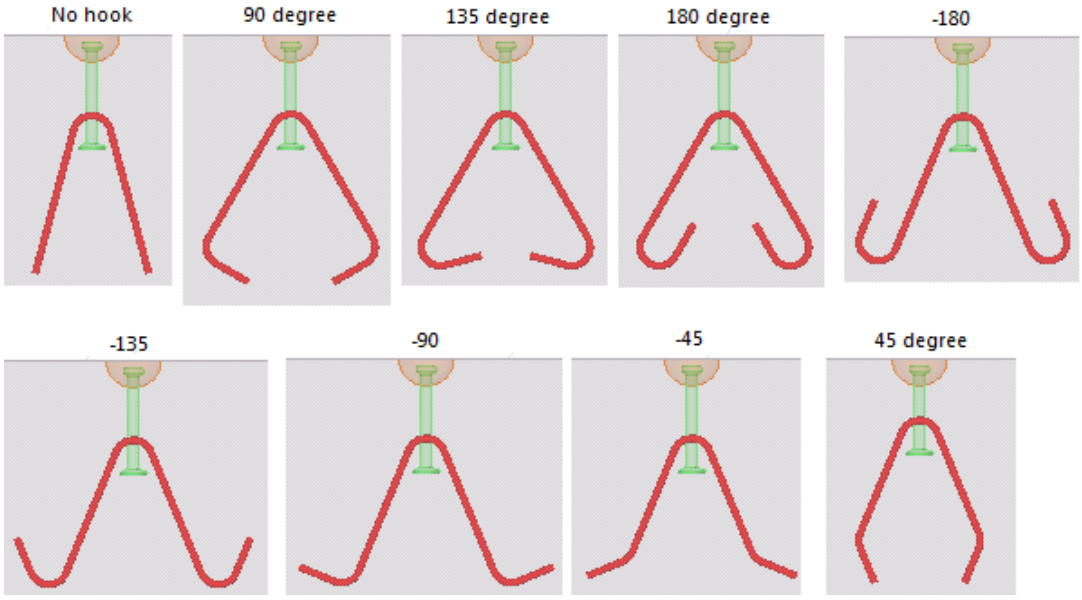


Grade, Size

The **Grade** and **Size** options work together. Click the **...** button next to the **Size** box to open the reinforcing bar catalog and to select a grade and a size.

End conditions left/End conditions right

Select the shape of the reinforcing bar.

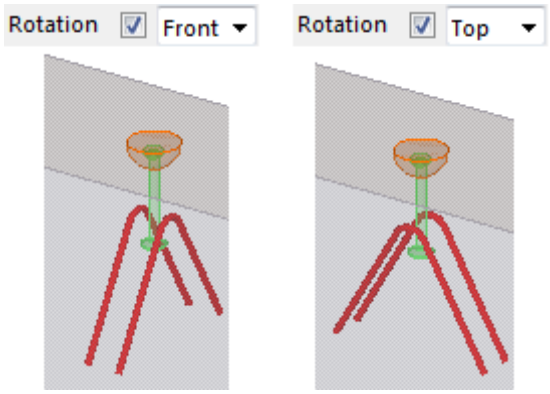


Bend lengths left/Bend lengths right, Bend radius

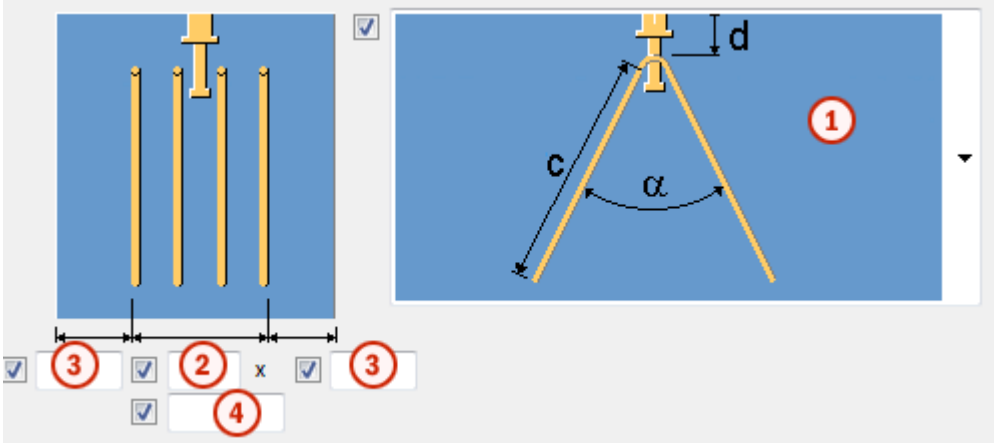
Define the bend length for the hooks and the bend radius.

Rotation

Select how to rotate the hangup reinforcing bars and define the rotation angle.



Dimensions



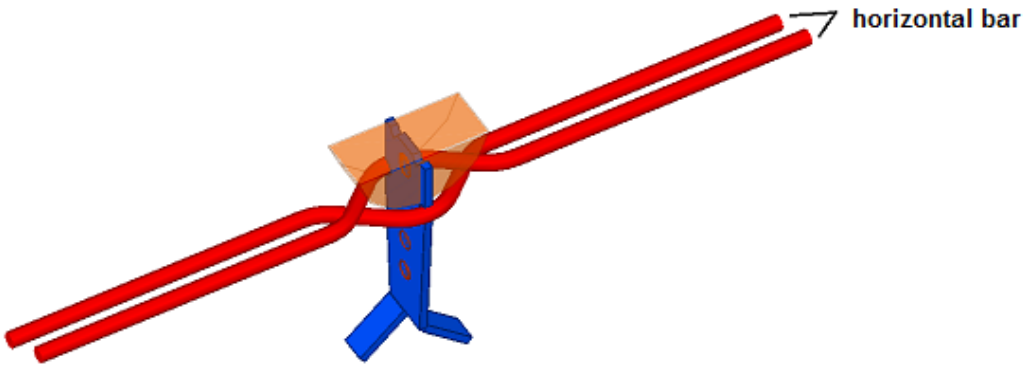
Option	Description	Default
1	Select how to place the hangup reinforcing bars. Use the values **a**, **b**, **c**, **d**, **e**, **f** and the angles **alpha** and **beta** to define the dimensions of the hangup reinforcing bars.	
2	Define the number of hangup reinforcing bars.	0
3	Define the cover thickness.	
4	Define the distance between hangup reinforcing bars.	100 mm

Horizontal bar tab

Use the **Horizontal bar** tab to define additional horizontal reinforcing bars for the embeds.

You can define the reinforcing bar shape and hook properties, and the reinforcing bar profile properties.

Example



Grade, Size

The **Grade** and **Size** options work together. Click the **...** button next to the **Size** box to open the reinforcing bar catalog and to select a grade and a size.

End conditions left/End conditions right

Select the shape of the reinforcing bar.

Bend lengths left/Bend lengths right, Bend radius

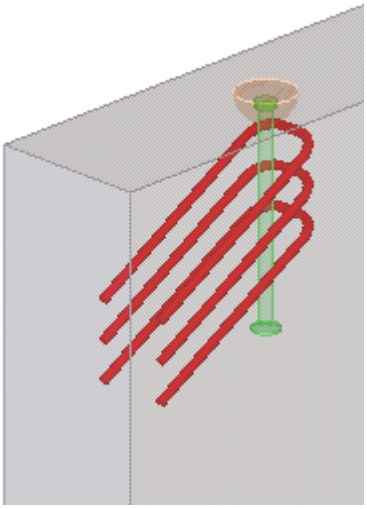
Define the bend length for the hooks and the bend radius.

Rotation

Select how to rotate the horizontal reinforcing bars and define the rotation angle.

Side view

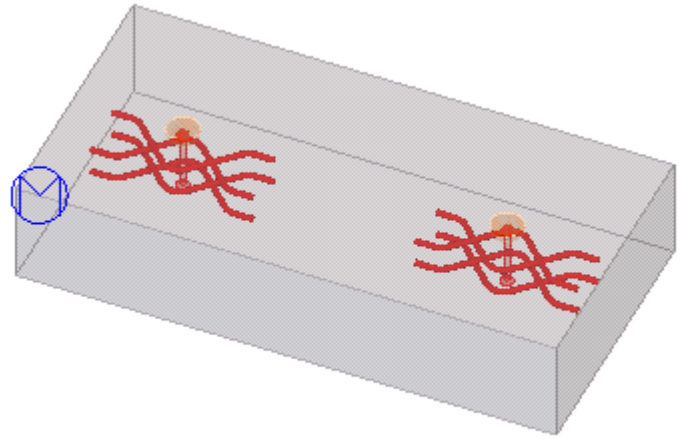
You can create sloped horizontal reinforcing bars. Define the number of bars and the distance between them. Use value **d** to define cover thickness for the top side and value **f** to define the reinforcing bar length.

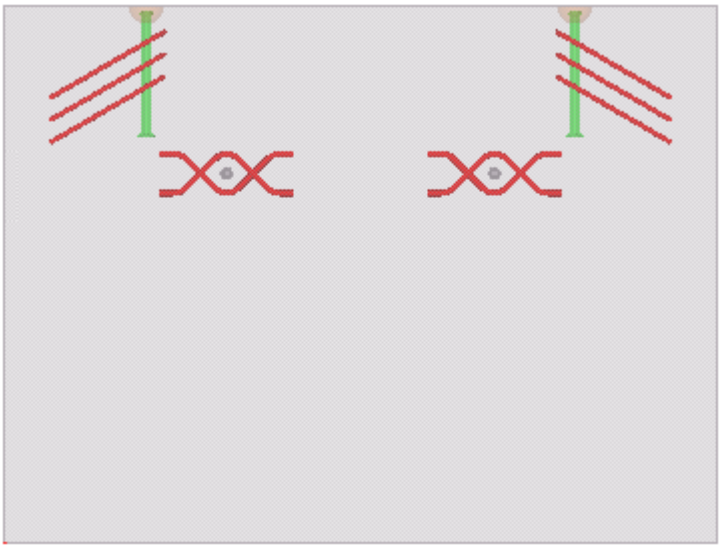


Top view

Select a reinforcing bar shape.

Use values **a**, **b**, **c**, **d**, **e**, **f** and the angle **α** to define the dimensions of the horizontal reinforcing bars.

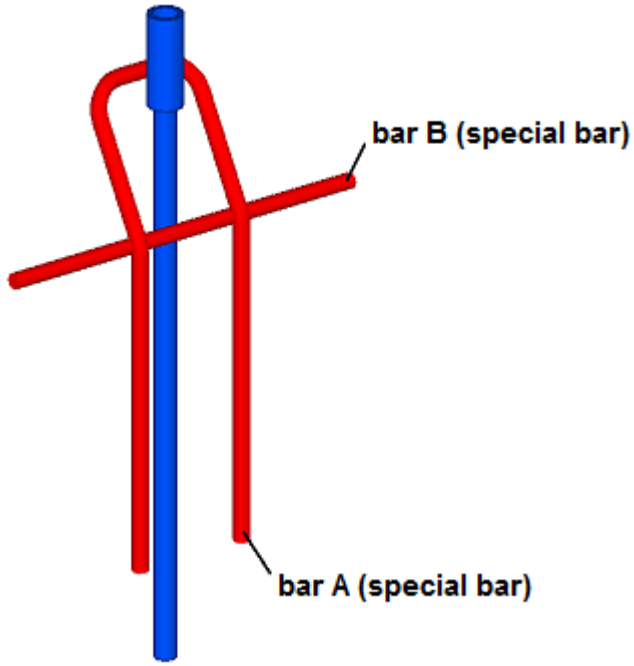




Special bar tab
Use the **Special bar** tab to define additional U-shaped reinforcing bars for the embeds.

You can define the reinforcing bar shape and hook properties, and the reinforcing bar profile properties in two directions.

Example



Grade, Size

The **Grade** and **Size** options work together. Click the **...** button next to the **Size** box to open the reinforcing bar catalog and to select a grade and a size for **bar A** and **bar B**.

End conditions left/End conditions right

Select the shape of the reinforcing bar.

Bend lengths left/Bend lengths right, Bend radius

Define the bend length for the hooks and the bend radius.

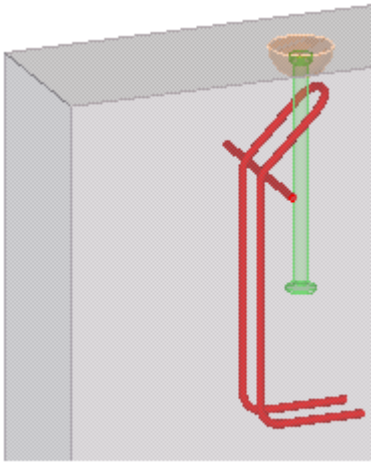
Rotation

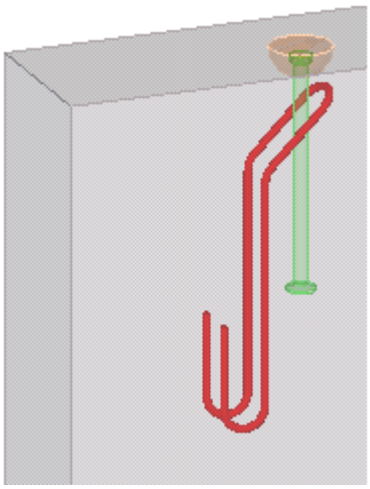
Select how to rotate the U-shaped reinforcing bars and define the rotation angle.

Shape

Select a reinforcing bar shape.

Use the values **a**, **b**, **c**, **d**, **e**, **f** and the angles **α** to define the dimensions of the U-shaped reinforcing bars.



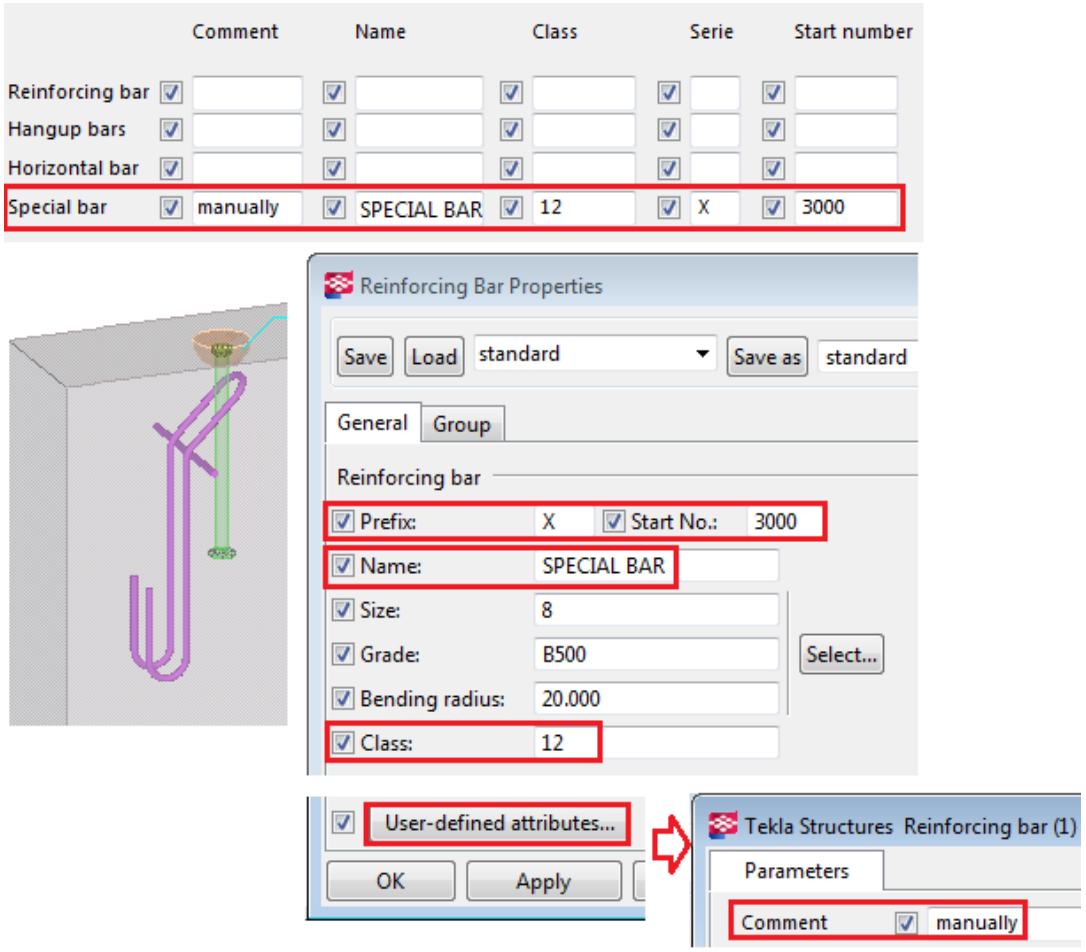


Advanced tab

Use the **Advanced** tab to define UDAs and reinforcing bar properties for the reinforcing bars, hangup bars, special bars and horizontal bars.

Reinforcing bar properties

Option	Description
Comment	Add a comment for the reinforcing bars.
Name	Define a name that is shown in drawings and reports.
Class	Define the class number for the reinforcing bars.
Serie	Define a prefix for the bar position number.
Start number	Define a start number for the bar position number.



Create as rebar assembly

You can add the created reinforcement as a rebar assembly to the cast units with a predefined assembly type, name, prefix, and start number.

Option	Description
Create as rebar assembly	Select **Yes** to create all reinforcement as one rebar assembly, and to include it to the cast unit of the input part.
Add to existing rebar assembly	• **Do not add**: New rebar is added as a rebar assembly to the cast unit. • **As single bars**: Add all new rebars directly into an existing rebar assembly as rebars. • **As sub-assembly**: Add all new rebars to their own new rebar assembly, which is then included

Option	Description
	as a sub-assembly to the main rebar assembly.
Rebar assembly type	Select the rebar assembly type. If you do not select the type, the default value of the rebar assembly is used.
Name, **Profile**, **Start number**	Define the name, profile, and start number. If you do not define these, the default values of the rebar assembly are used.

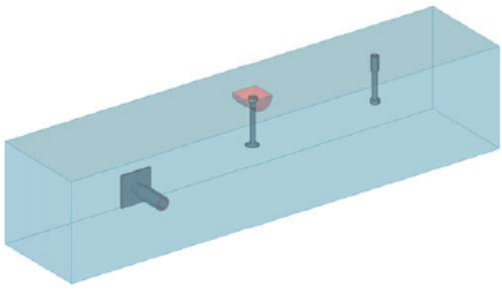
Embed (1008)

Embed (1008) creates one or more embeds in concrete parts. You can create multiple embeds to be used as lifting anchors with one insertion point.

Objects created

- Embeds
- Reinforcing bars

Use for

Situation	Description
	Embeds

Selection order

1. Select a concrete part.
2. Pick one point on the part face where you want to insert the embed.

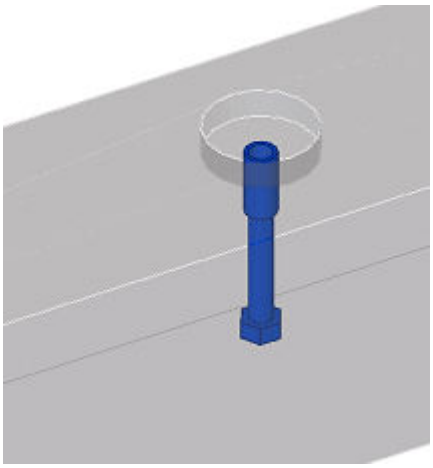
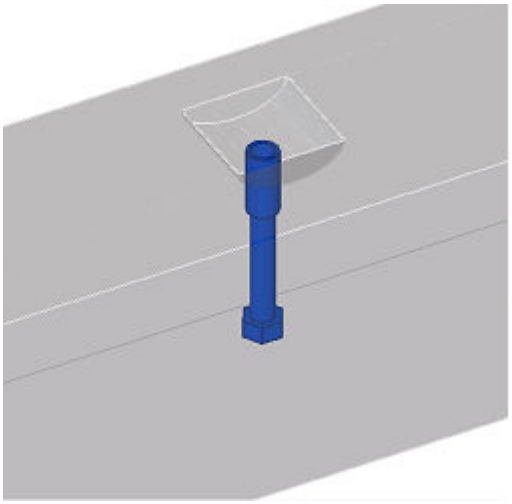
   The detail is created automatically when you pick the point.

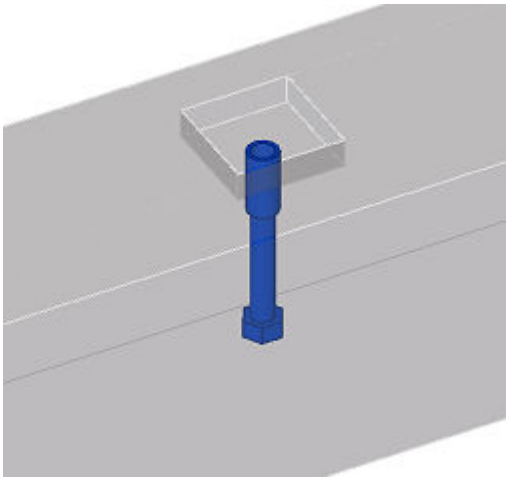
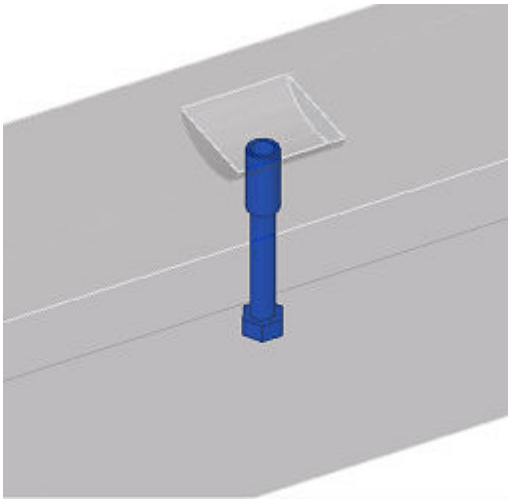
Picture tab

Use the **Picture** tab to control the embed dimensions, connection method and rotation.

Embed dimensions

Define if the embed needs to be recessed. You can define the dimensions of the recess, distance from the recess plane to the top of the embed, select the shape of the cutout and whether the cutouts are handled as empty cutouts or cutouts with a formwork part.

Option	Example
Circle	
Half moon X	

Option	Example
Square	
Half moon Y	

Top part/Bottom part

Set the part class and rotation. Each option rotates the embed 90 degrees counterclockwise. You can define also a fixed rotation angle.

How to connect top part to concrete element

Select whether the top part of the embed is created, and if set to **Yes**, select how the part is connected to the concrete part.

How to connect bottom part

Select whether the bottom part of the embed is created, and if set to **Yes**, select how the part is connected to the concrete part.

Select a custom part from the Applications & components catalog

Select a custom part from the **Applications & components** catalog to be used as embed. To use saved custom component properties, select the saved properties file.

Use the **Yes and subassembly** option to add the embed as a subassembly to the main part. The default direction is **2 point +x**.

Top part tab

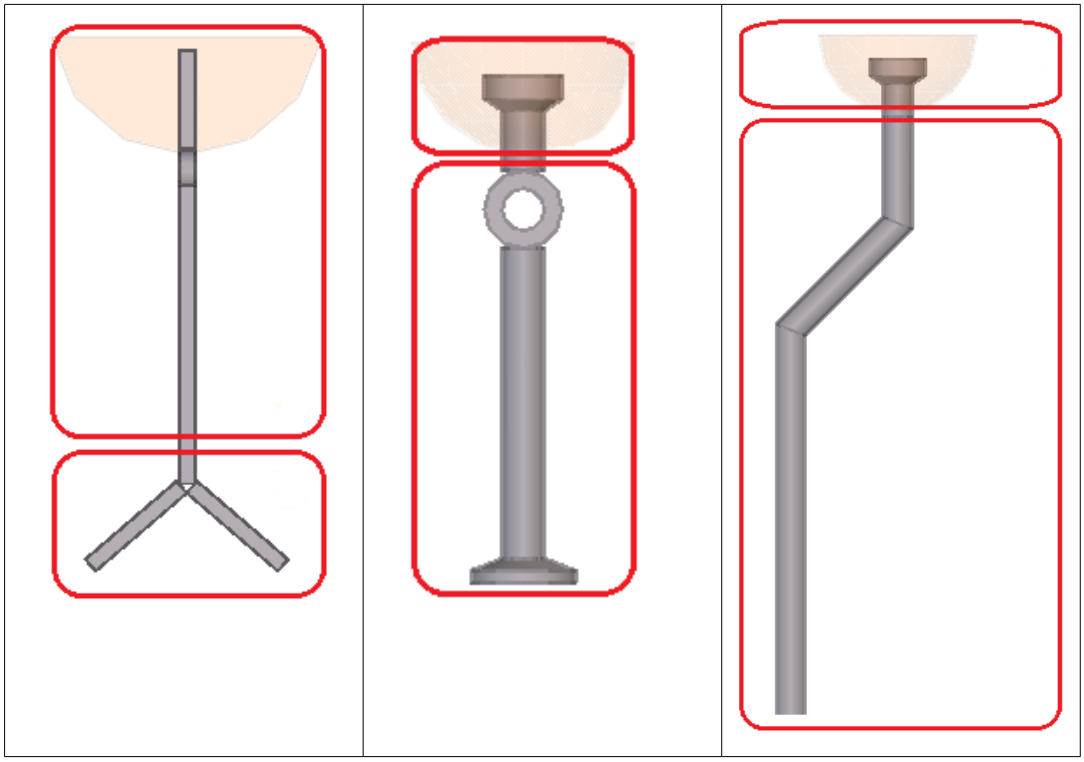
Use the **Top part** tab to define the top part of the embed.

Top part dimensions

Use the diameter and height boxes to define the shape of the top part of the embed. You can define the main diameters of parts **1**, **2**, **3**, **4** and **5** also on the **Parts** tab.

Examples

Embed top part defined on the **Top part** tab, embed bottom part defined on the **Bottom part** tab.



Bottom part tab

Use the **Bottom part** tab to define the bottom part of the embed.

Bottom part dimensions

Use the diameter and height boxes to define the shape of the bottom part of the embed. You can define the main diameters of parts **1**, **2**, **3**, **4** and **5** also on the **Parts** tab.

For examples, see **Top part** tab.

Parts tab

Use the **Parts** tab to define the embed top and bottom part profiles and the formwork part properties.

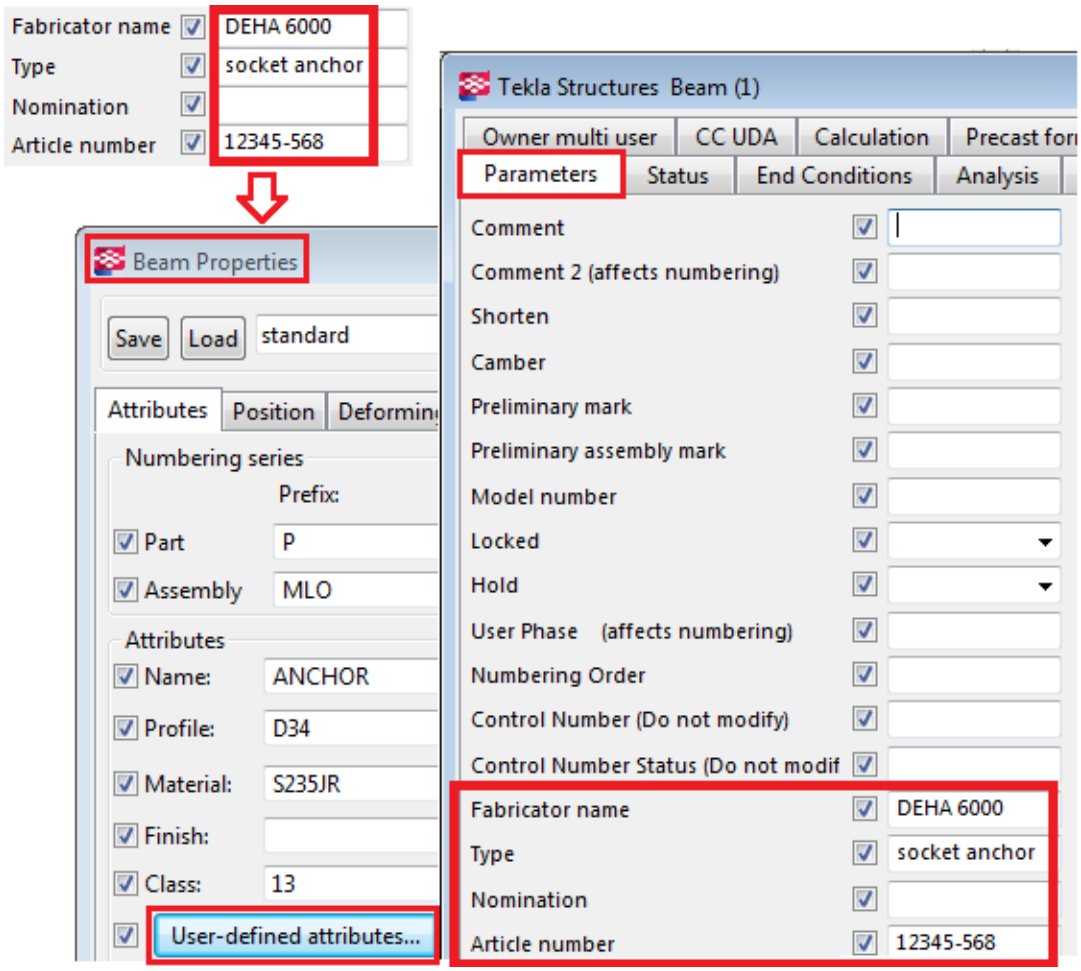
Part properties

Both the top and the bottom part are build from multiple profiles. You can define profiles for each section.

Define the part properties for the top, bottom and formwork part. If the profile properties are left empty, the lengths and diameters defined on the **Top Part** and **Bottom Part** tabs are used.

Option	Description
t, **b**, **h**	Define the part thickness, width and height.
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name for the part.
Comment	Add a comment for the part.

You can define UDAs for the top and bottom parts.



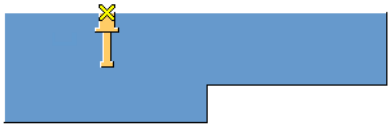
Placement tab
Use the **Placement** tab to define the embed placing, positioning and embed distribution.

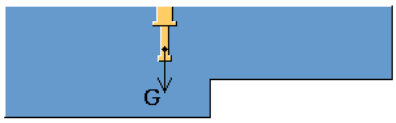
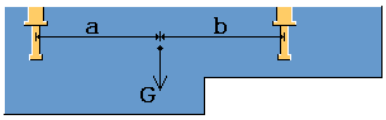
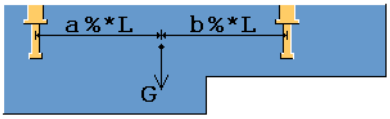
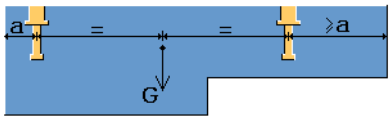
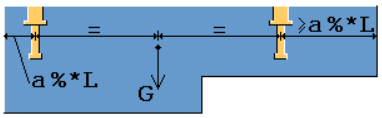
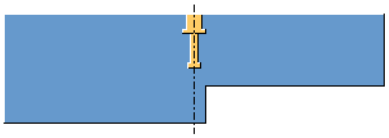
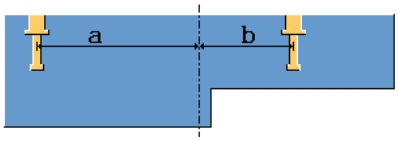
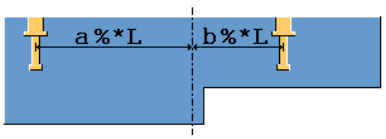
Positioning

Select how to position the top and the bottom part of the embed.

Placing

Define the placing type of the embed. Depending on the component insertion points, multiple embeds can be created.

Option	Description
	Embed is placed on a selected position point.

Option	Description
	Embed is placed on a center of gravity (COG) point in the part length direction.
	Multiple embeds. Define dimensions **a** and **b**. Reference = COG
	Define dimensions **a** and **b** as percentages of the part length. Reference = COG
	Embed is placed in the middle of the part, along the length of the concrete part.
	Multiple embeds. Define dimension **a** as a percentage of the total part length. Reference = COG
	Embed is placed in the middle of the part.
	Define dimensions **a** and **b**. Reference = middle of the part
	Define dimensions **a** and **b** as percentages of the part length. Reference = middle of the part
	Define dimensions **a** and **b**. The distances are from the embeds to the part ends.
	Define dimensions **a** and **b** as percentages of the part length. The distances are from the embeds to the part ends.

Dimensions

Define embed dimensions **a** and **b**.

Center of gravity

Define concrete part COG (center of gravity) for the embeds.

Number of extra anchors

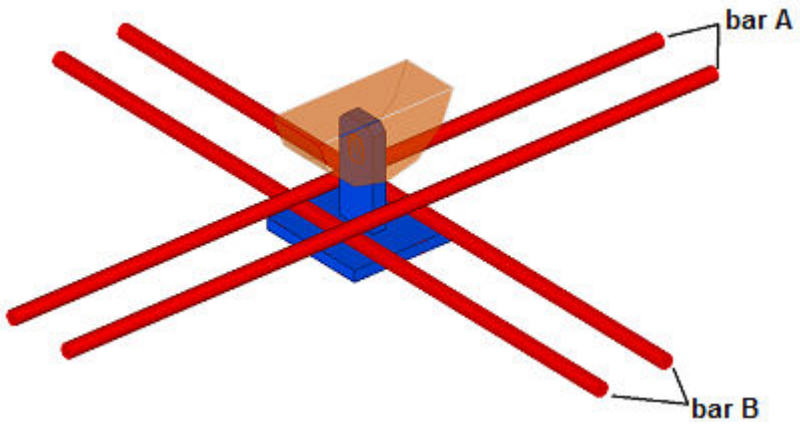
The spacing distance between multiple embeds.

Reinforcing bar tab

Use the **Reinforcing bar** tab to define extra reinforcing bars for the embeds.

You can define the reinforcing bar shape properties, and the reinforcing bar profile properties in two directions.

You can define the number, shape, dimension and covering thickness of the reinforcing bars.



Advanced tab

Use the **Advanced** tab to define bar properties for reinforcing bars A and B.

Reinforcing bar properties

Option	Description
Comment	Add a comment for the reinforcing bars.
Name	Define a name that is shown in drawings and reports.
Class	Define the part class number for the reinforcing bars.
Serie	Define a prefix for the part position number.
Start number	Define a start number for the part position number.

Create as rebar assembly

You can add the created reinforcement as a rebar assembly to the cast units with a predefined assembly type, name, prefix, and start number.

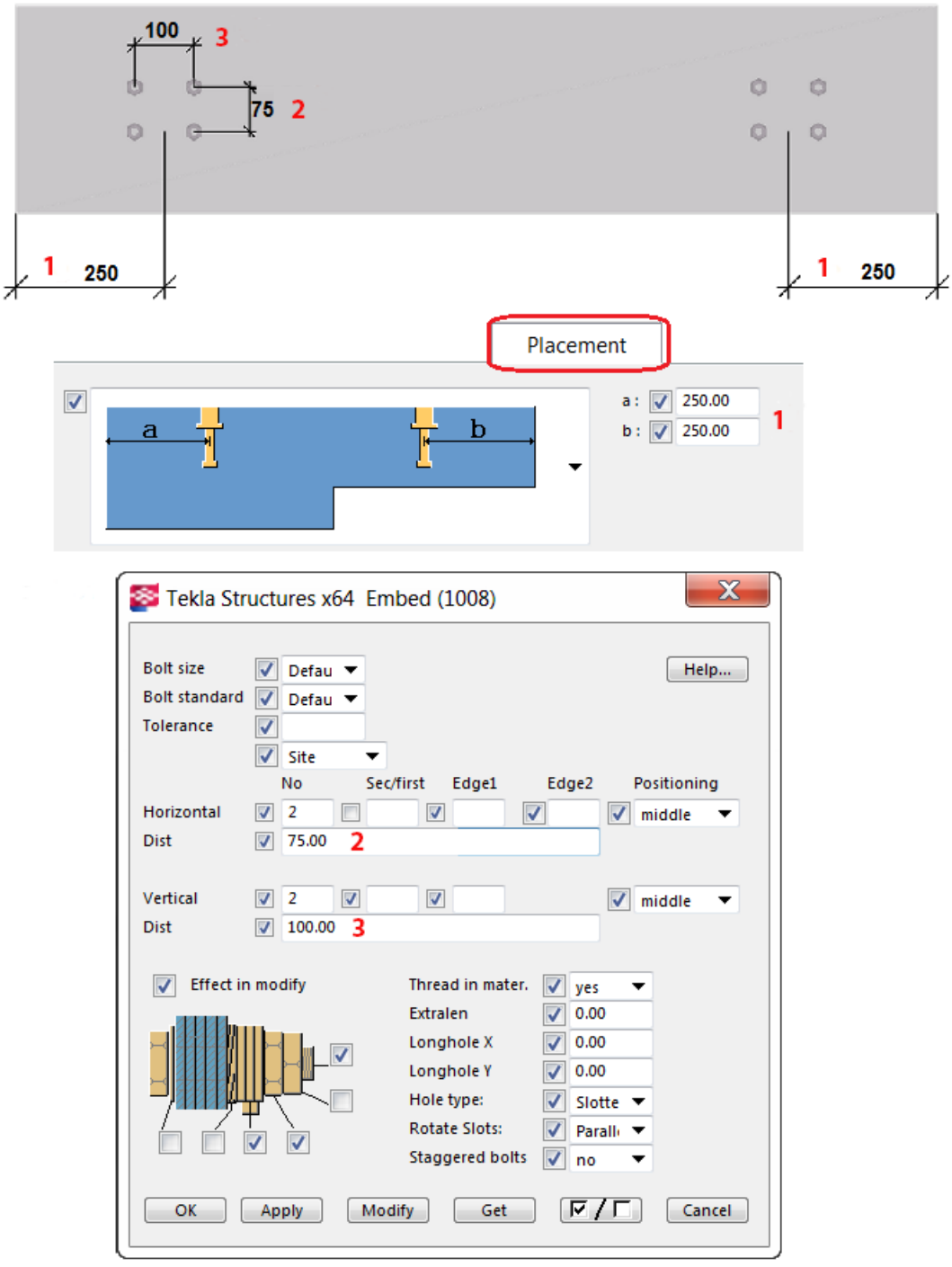
Option	Description
Create as rebar assembly	Select **Yes** to create all reinforcement as one rebar assembly, and to include it to the cast unit of the input part.
Add to existing rebar assembly	• **Do not add**: New rebar is added as a rebar assembly to the cast unit. • **As single bars**: Add all new rebars directly into an existing rebar assembly as rebars. • **As sub-assembly**: Add all new rebars to their own new rebar assembly, which is then included as a sub-assembly to the main rebar assembly.
Rebar assembly type	Select the rebar assembly type. If you do not select the type, the default value of the rebar assembly is used.
Name, **Profile**, **Start number**	Define the name, profile, and start number. If you do not define these, the default values of the rebar assembly are used.

Bolts
Click the **Bolts** button to open the **Bolts** dialog box where you can define extra embeds and embed offsets.

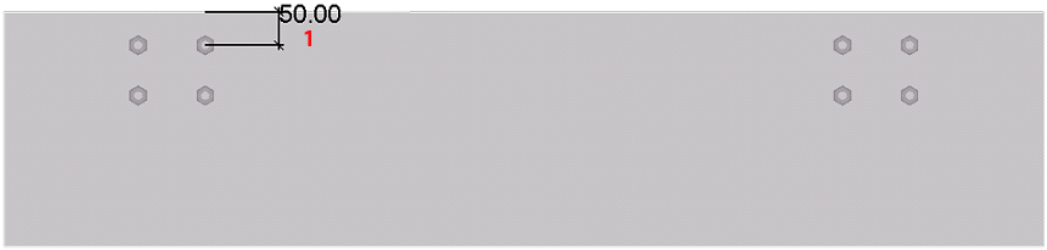
Embed (1008) uses only the **Horizontal Dist.** and **Vertical Dist.** options, the other options are not taken into account.

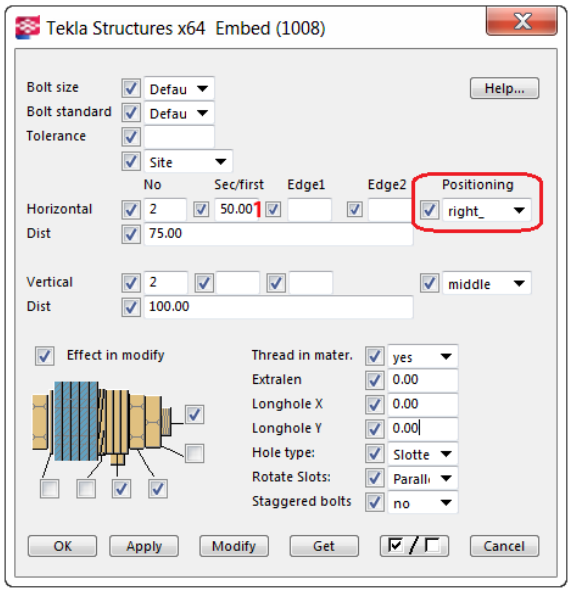
Example

In the example below, the bolt placement uses fixed dimensions from part edges, defined on the **Placement** tab. Extra embeds are defined in the **Bolts** dialog box.



Optionally, you can also use the **Sec/first** and **Positioning** options to define a distance from the edge of the beam.





General tab

Click the link below to find out more:

General tab

Analysis tab

Click the link below to find out more:

Analysis tab

Continuous Beam Reinforcement

You can reinforce a continuous beam using a macro called **ContinuousBeamReinforcement**. The macro creates main top and bottom bars, stirrups, fittings, and additional top and bottom bars using system components. **Longitudinal reinforcement (70)** creates the main and

additional top and bottom bars, **Stirrup reinforcement (67)** creates stirrups, and **Battering connection (13)** creates fittings.

To reinforce a continuous beam:

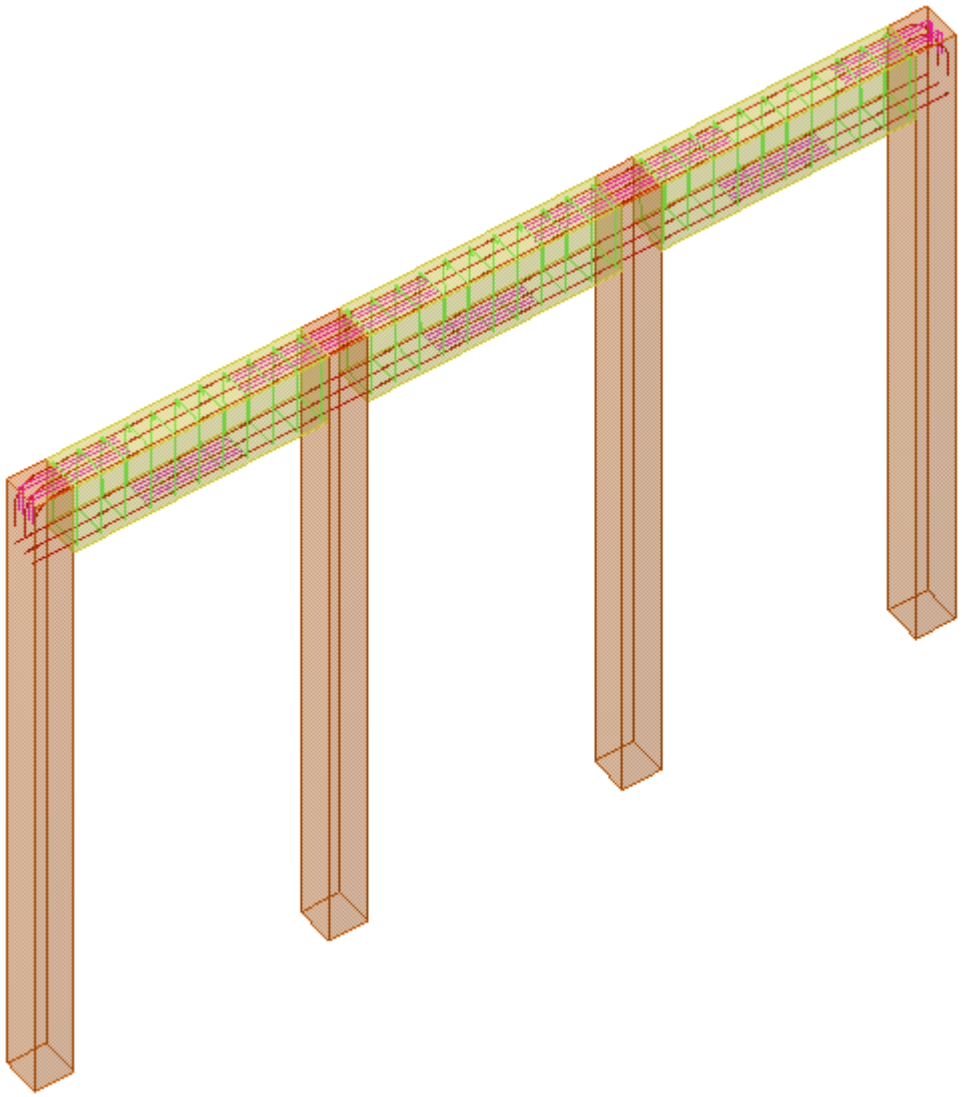
1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. Click the arrow next to **Applications** to open the applications list.
3. Double-click **ContinuousBeamReinforcement**.
4. Select the beams and click **Next**.
5. Select the columns and click **Next**.
6. If needed, modify the reinforcement properties, and click **Finish**.

   The reinforcements are created.

Limitations

The beams must be aligned for the reinforcing to be successful.

Example



Lifting

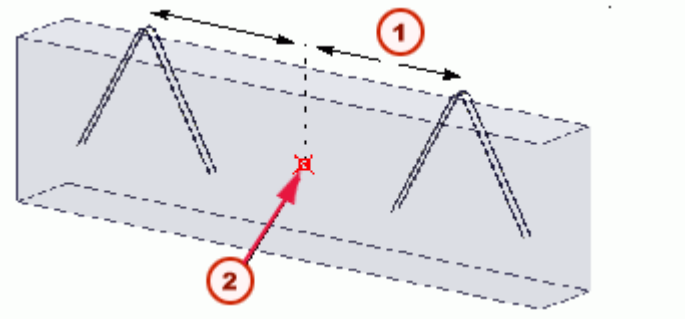
This section introduces components that can be used for lifting.

Click the links below to find out more:

- Lifting anchor (80) (page 4082)

Lifting anchor (80)

Lifting anchor (80) creates two lifting anchors (or anchor groups) for a concrete part and places them symmetrically on either side of the part's center of gravity.

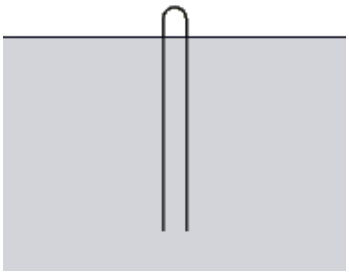
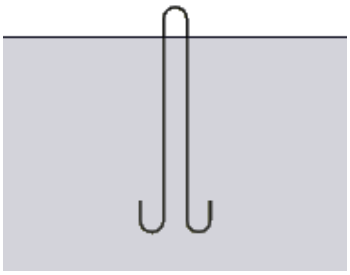
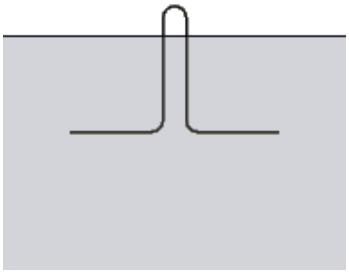
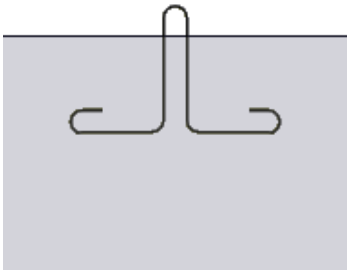


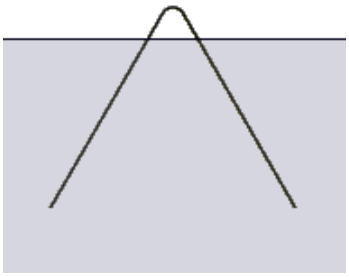
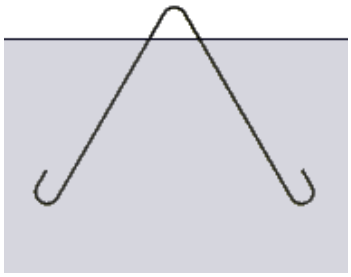
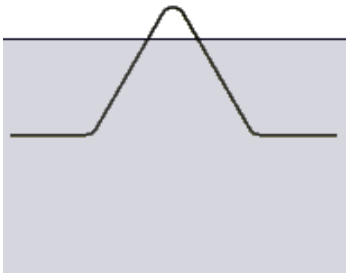
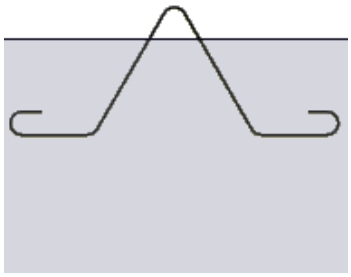
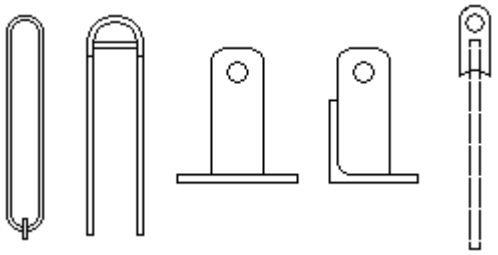
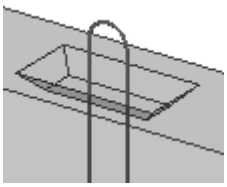
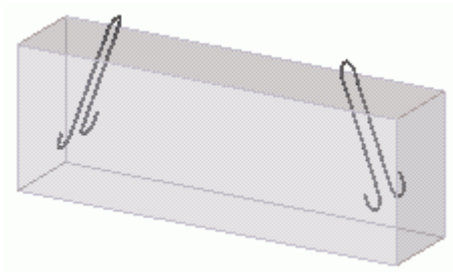
	Description
1	30% of part length (default)
2	Center of gravity

Objects created

- Lifting anchors (2 or more)
- Recesses for anchors (optional)

Use for

Situation		More information
		Straight anchor with straight legs (Type A in the anchor properties file, see **Anchor properties from a file**)
		Straight anchor with L-shaped legs (Type D)

Situation	More information
 	Angle anchor with straight legs (Type B)
 	Angle anchor with L-shaped legs (Type C)
	Custom components as anchors
	Anchors recessed into the part.
	Skew and/or rotated anchors

Before you start

Create the concrete part.

Selection order

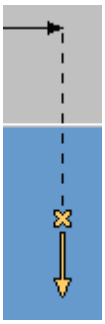
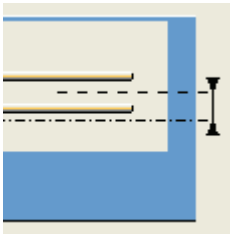
1. Concrete part.

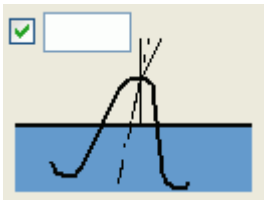
Picture tab

Use the **Picture** tab to define anchor properties, to use custom components as anchors, to create recess and define recess dimensions.

Lifting anchor properties

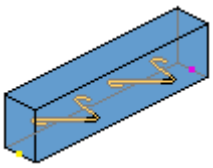
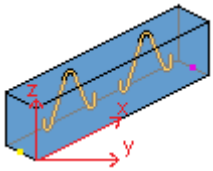
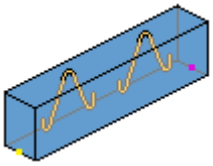
The properties of lifting anchors are:

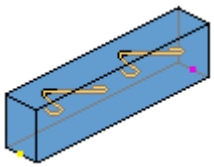
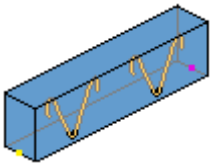
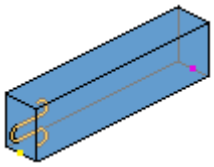
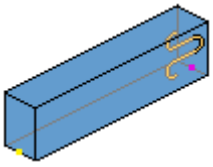
Option	Description
Create anchors	Select how the anchors are created.
Anchor dimensions	The options are: • **Use dialog values** • **Select from file**, see **Anchor properties from file**. You need to enter different dimensions for different anchor type
Distance from	Set the anchor distribution from the center of gravity or part enc You can set the distance either by length or by percentage of the
	The distance between an anchor (or anchor group) and the cente of the part. Default is 30% of the part length.
	The distance between an anchor (or group) and the center line o
Number of bars/spacings	The number and spacing of anchors in a group. If the spacing va each value individually.
Custom	See **Custom component as lifting anchor**.
Component	
Custom settings	
Up direction	

Option	Description
	The option to define if the anchors are recessed into the part.
	The skew angle of anchors. Anchor heads are skewed towards th center of gravity.
Rotate anchor	The option to rotate anchors. The options are: • **No**: Anchors parallel to the part. • **Yes**: Anchors perpendicular to the part.
Grade	Strength of the steel used in the anchor.
Size	Diameter of the anchor bar.
Name	Name that is shown in drawings and reports.
Prefix, **Start number**	Prefix and start number for the part position number.
Class	Part class number.

Side

Select the side of the part where Tekla Structures creates the anchors:

Option	Description
	Front
 	Top Default

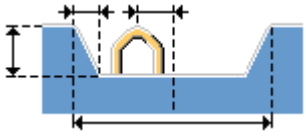
Option	Description
	Back
	Below
	Start end
	Finish end

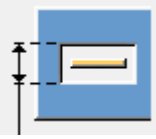
Recesses

Select one of the following options to define if the lifting anchors are recessed into the concrete part:

Option	Description
	Anchors on the surface of the part. Default
	Anchors recessed into the part.

If you choose to recess the anchors into the part, use the following fields to define the shape and dimensions of the recesses:





Custom component as lifting anchor

To use custom components as lifting anchors:

1. In the **Custom** list box, select **Yes**.
2. Click the **...** button next to the **Component** field to open the **Select component** dialog box.
3. Browse for the custom component you want to use as lifting anchor.

   The component you select must be a custom part and have two or three input points.

   The component should be created so that the first and the second input points are on the concrete surface and along the longitudinal axis of the concrete part.

   TIP For more information on how to create custom parts that can be used as lifting anchors, see Creating standard embeds for global use.

4. Select the component and click **OK**.
5. To use saved custom component properties, select the saved properties file in **Custom settings**.
6. If the anchor position is not correct, select another option in the **Up direction** list box.

Anchor properties from file

You can define lifting anchor properties by entering values in the dialog box, or you can create a file containing the anchor properties you want to use. Use any standard text editor to create the file and save it as `LiftingAnchors.dat` in the model folder.

To use the anchor properties you define in a file:

1. In the **Lifting anchor (80)** dialog box, select **Select from file** from the **Anchor dimensions** list box.
2. Click the **...** button to browse for the file.

Example

Create a row in the file for each lifting anchor. Enter the following properties, separated by spaces:

- Anchor capacity [kN], including safety and material factors

- Type [A, B, C, or D], see the **Use for** table.
- Grade [characters]
- Size [characters]
- Anchoring length 1 [mm]
- Anchoring length 2 [mm]
- Protruding dimension of the anchor outside the part [mm]
- Penetrating dimension of the anchor inside the part [mm]
- Bending radius [mm]
- Hook length [mm] (0 = no hook)
- Angle of leg 1 [degrees]
- Angle of leg 2 [degrees]

Here is an example of a lifting anchor file (Tekla Structures ignores the comments enclosed in /* */):

LiftingAnchors.dat

```
10 B A500HW 10 400 400 100   0 30  75 30 30 /*      /\       */
20 B A500HW 12 600 600 150   0 36 100 30 30 /*     /  \      */
30 B A500HW 16 900 900 200   0 80 150 30 30 /*    \/  \/     */
10 B A500HW 10 400 400 100   0 30   0 30 30 /*     /\       */
20 B A500HW 12 600 600 150   0 36   0 30 30 /*    /  \      */
30 B A500HW 16 900 900 200   0 80   0 30 30 /*    /  \      */
10 C A500HW 10 400 400 100 200 30  75 30 30 /*     /\       */
20 C A500HW 12 600 600 150 300 36 100 30 30 /*  _ /  \ _    */
30 C A500HW 16 900 900 200 500 80 150 30 30 /* |__/    \__| */
10 C A500HW 10 400 400 100 200 30   0 30 30 /*     /\       */
20 C A500HW 12 600 600 150 300 36   0 30 30 /*    /  \      */
30 C A500HW 16 900 900 200 500 80   0 30 30 /* ___/    \___ */
```

Tekla Structures will use the first anchor in the file that:

1. Has the shape and type you define using the graphic list boxes in the **Lifting anchor (80)** dialog box.
2. Can carry the part's weight with other identical anchors. The total number of anchors is two times the number you specify in the **Number of bars** field.

6.3 Automated Reinforcement Layout

This section explains how to create reinforcement using the automated tools available in Tekla Structures.

Click the links below to find out more:

- Double-tee beam (51) (page 4090)

- Inverted-tee beam (52) (page 4110)
- Ledge beam (53) (page 4121)
- Rectangular beam (54) (page 4132)
- Ledge spandrel (55) (page 4141)
- Rectangular spandrel (56) (page 4151)
- Columns (57) (page 4159)
- Topping (58) (page 4174)
- Wash (59) (page 4175)
- Hollowcore reinforcement strands (60) (page 4178)
- Post-tensioning (61) (page 4181)
- Edge and Corner Reinforcement (62) (page 4184)
- U Bar of concrete slab (63) (page 4190)
- Wall panel (64) (page 4192)
- Reinforcement Strand Layout (66) (page 4204)

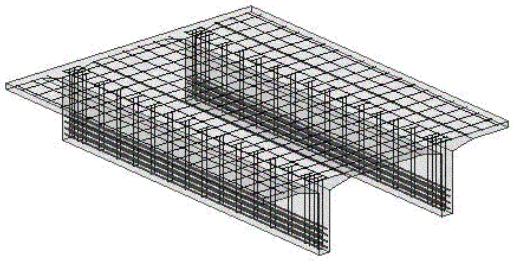
Double-tee beam (51)

Double-tee beam (51) creates flexural, shear, and flange reinforcement for a double-tee beam.

Objects created

Reinforcement meshes.

Use for

Situation	Description
	Double-tee with reinforcement.

- Prestress straight or depressed strands for flexural reinforcement in double-tee stems
- Mild longitudinal bars for flexural reinforcement in double-tee stems
- Mesh for shear reinforcement in double-tee stems
- Stirrups in double-tee stems

- Reinforcement mesh in the double-tee flange
- Transverse and longitudinal mild reinforcing bar in the ends and sides of the double-tee flange
- Reinforcement around openings and notches in the double-tee flange

Before you start

- Create a concrete double-tee beam.
- Calculate the required area and quantity of strands and other reinforcement.
- Define other reinforcement properties, such as debonding and pull values.

Selection order

1. Select the double-tee beam.

   The reinforcement is created automatically.

Strand template tab

Use the **Strand template** tab to define the number and arrangement of vertical strand planes in each stem, and the vertical distances between grid points in each plane.

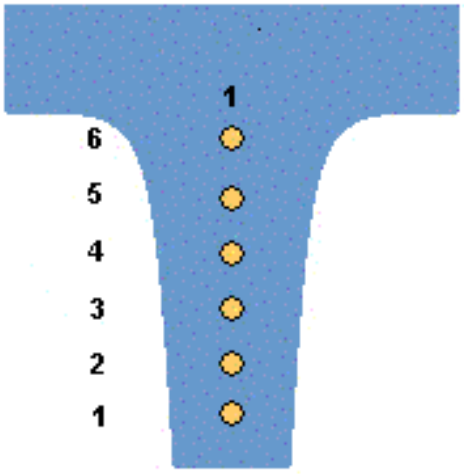
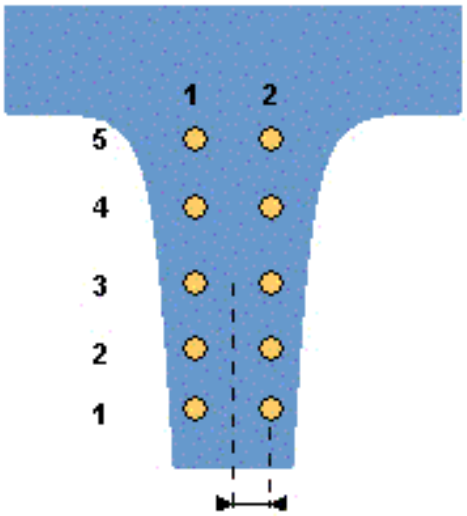
Grid spacing along Y and along X

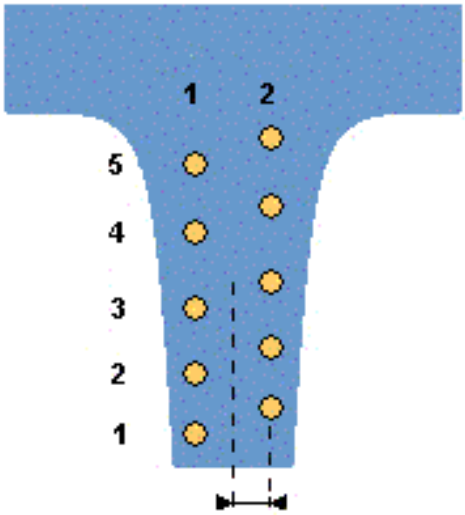
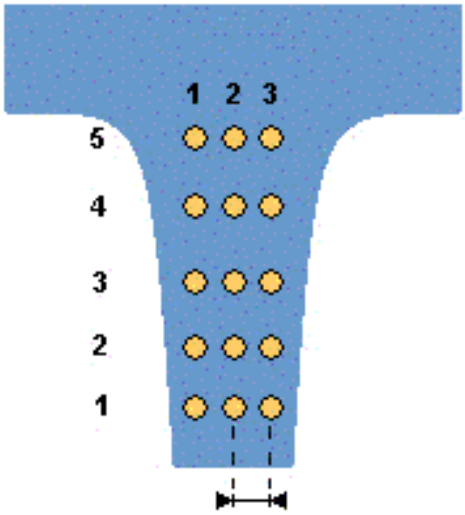
Define the grid spacing values for the y axis. If you have selected the strand plane **Triple**, also define grid spacing values for the x axis.

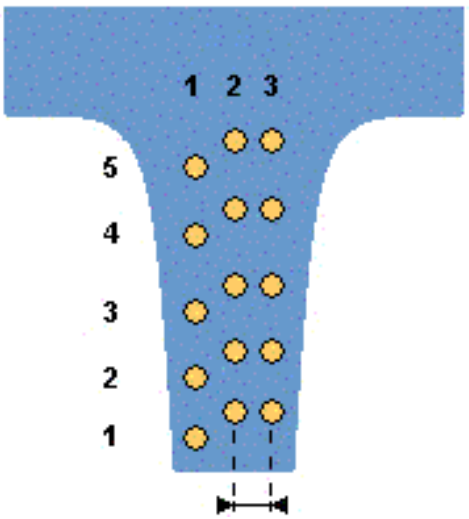
Separate the values with commas or spaces. The first value is the distance from the the bottom edge of the stem. Each successive value is the distance to the next grid point.

TIP Use a multiplier to define a repeating series of values. For example, 8*2 indicates eight holes with a two unit spacing.

Strand planes

Option	Description
	Single
	Double

Option	Description
	Double staggered
	Triple

Option	Description
	Triple staggered

Strand pattern tab

Use the **Strand pattern** tab to define the placement of strands in the grid points.

External file

With an external CSV file you can define an unlimited amount of individual strands.

Select to use an external file from the **Use external file** list.

Select an external CSV file by clicking **...** button.

Rebar end is same as start

Define the strand input.

- Select **Yes** to use the row and debond information of the strand start for the strands at the strand end.
- Select **No** to define an alternate row and debond information at the end of each strand.

Strand pattern definitions

Define the row and column for each end of the strand using the strand template positions in the strand grid. Strands may have different grid row positions at each end, but a strand cannot cross grid columns.

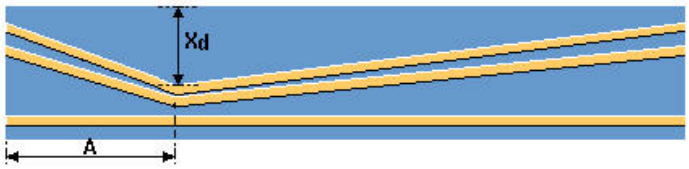
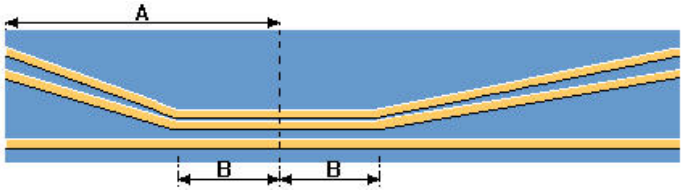
Option	Description
Strand start - Row	Define the horizontal row number in the strand template for the start of the strand.

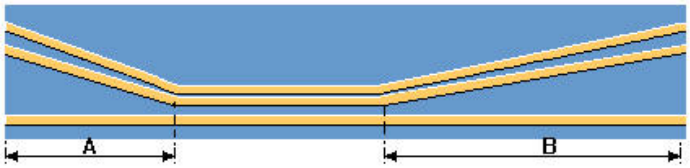
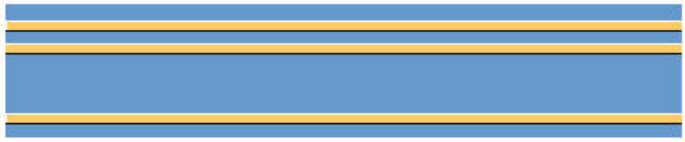
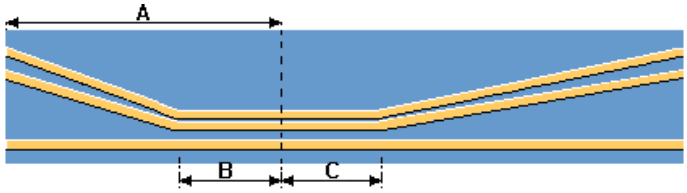
Option	Description
Strand start - Debond	Define the debond length at the start of the strand.
Strand end - Row	If **Same as left** is **No**, define the horizontal row number in the strand template for the end of the strand.
Strand end - Debond	If **Same as left** is **No**, define the debond length at the end of the strand.
Columns	Define a list of one or more column numbers and/or column ranges in the strand template at which a strand should be created.
Size	Define the size of the strands by selecting if from the reinforcing bar catalog.
Grade	Define the strand grade.
Pull	Define the pull force of the strands.
Create strands in	Select strand pattern side settings: • **Both stems** Default • **Right stem** • **Left stem**

Strand profile tab

Use the **Strand profile** tab to define profiles for the strands. All strands in each stem use the same general profile.

Strand depression profiles

Option	Description
	One point, depressed
	Two points, depressed variable

Option	Description
	Two points, depressed fixed
	Straight
	Three points, depressed variable

Depress position

Define depress positions **A**, **B**, and **C** as either a fixed distance or as a percentage of the member length.

Same as A and Same as B

Define whether depress position B uses the same values as depress position A in the **Two points, depressed variable** and **Two points, depressed fixed** strand profiles.

Define whether depress position C uses the same values as depress position B in the **Three points, depressed variable** strand profile.

Depression spacing type

To define the depression spacing type, select **One point, depressed** as the strand profile.

Option	Description
Exact location	Define the height of the strand in **Strand ht @ position**.
Top of depression	Define the distance **Xd** from the depress point in the outmost reinforcing bar to the edge of the stem.

Longitudinal rebar tab

Use the **Longitudinal rebar** tab to define longitudinal reinforcement.

Minimum end clear distance C & D

Define the minimum end clearance distance.

Longitudinal rebar definitions

Option	Description
Qty	Define the quantity of reinforcing bars.
Grade	Define the grade of the reinforcing bar.
Size	Define the size of the reinforcing bar by selecting it from the reinforcing bar catalog.
Ht	Define the distance from the end of the reinforcing bar to the face of the strand.

Length type

Define the length type for the reinforcing bar group. Enter additional information in the L, C, or D fields.

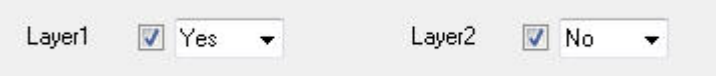
Option	Description
Full length	Reinforcing bar extends the full length of the stem, excluding the minimum end clear distance at each end. The L, C, and D fields are disabled.
Centered	Define a fixed length L for the reinforcing bar. The reinforcing bar is place at the center position in the stem. The C and D fields are disabled.
Known L, C	Define a fixed length L for the reinforcing bar and a distance C from the start of the strand. The D field is disabled.
Known L, C	Define a fixed length L for the reinforcing bar and a distance D from the end of the strand. The C field is disabled.
Known C, D	Define a distance C and D from the end of the strand. The L field is disabled.

Stem mesh tab

Use the **Stem mesh** tab to control the creation of stem meshes and their position.

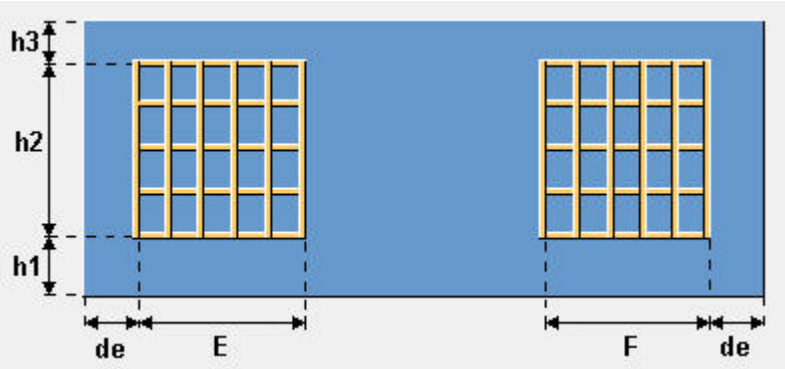
Stem mesh creation

Define whether one, two, or no stem meshes are created.



Stem mesh dimensions

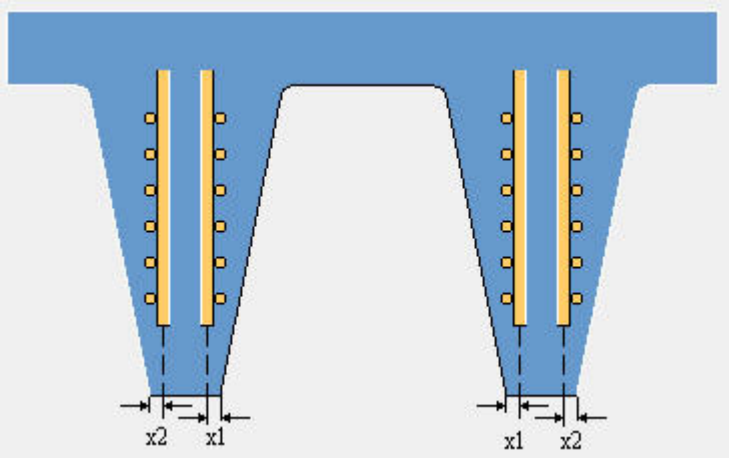
Define the mesh type by selecting it from the mesh type catalog. The mesh type also defines the mesh grade, mesh size, and bar spacing.



Option	Description
Mesh catalog type	Define the mesh type by selecting if from the mesh catalog.
End distance, de	Define the distance from the edge of the mesh to the edge of the stem.
Bottom clearance, h1	Define the distance from the edge of the mesh to the bottom edge of the stem.
Specify, h3	Select **Top clearance**. Define the distance from the edge of the mesh to the top edge of the stem.
Specify, h2	Select **Mesh height**. Define the height of the mesh.
Mesh full length	Define the mesh full length. The options are: • **Set length E** Define the left end length and right end length. • **Full length** • **Multiple sheets** creates overlapping meshes along the

Option	Description
	short and long sides of the stem. Define the overlap on the **Multiple sheets** tab.
Left end length, E	Define the length of the left end mesh.
Right end length, F	Define the length of the right end mesh.

Mesh stem position



Option	Description
Mesh stem position	Define the position for both ends of the stem, **x1** and **x2**.

Multiple sheets tab

Use the **Multiple sheets** tab to define the minimum overlap along the longitudinal and crossing direction of the meshes. Multiple sheets are created if you have selected **Mesh full length** to be **Multiple sheets** on the **Stem mesh** tab.

Overlapping

Option	Description
Min. overlapping (Short Side)	Define the minimum overlap along the crossing direction of the mesh.
Min. overlapping (Long side)	Define the minimum overlap along the longitudinal direction of the mesh.
Mesh leftover	Define whether the leftover is cut, **Cut leftover**, or not, **Overlap leftover**.

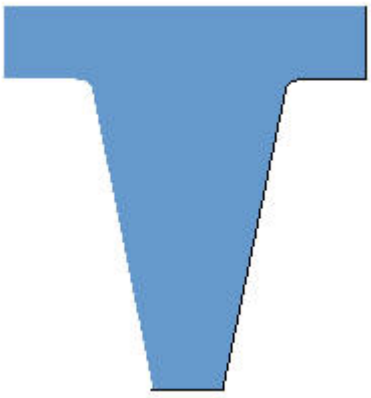
Stem stirrups tab

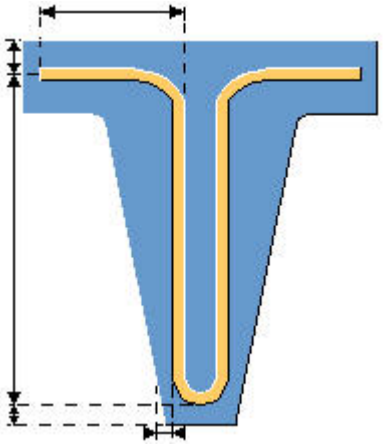
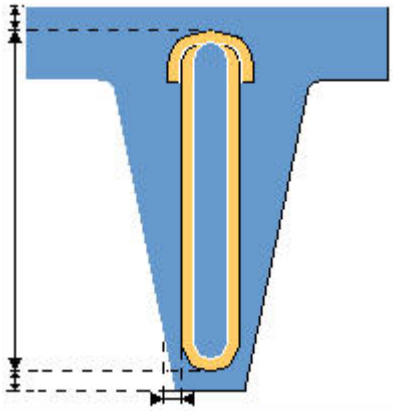
Use the **Stem stirrups** tab to define how the the stem stirrups are created.

Reinforcing bar properties

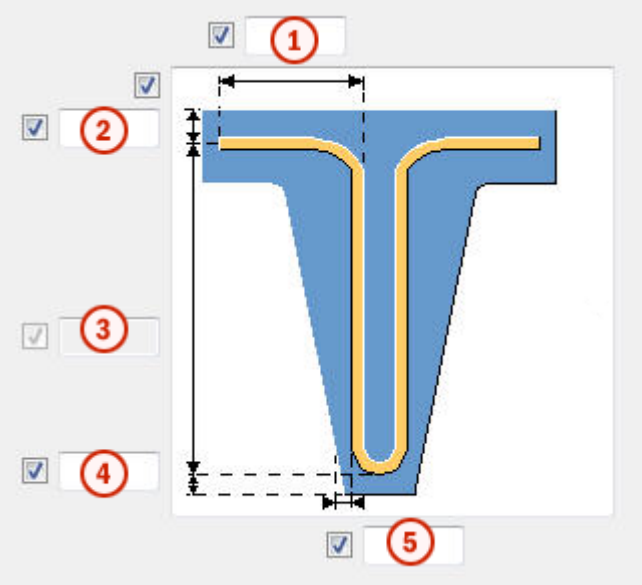
Option	Description
Spacing	Define the reinforcing bar spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.
Bars	Define the number of reinforcing bars.
Symmetrical	Define whether the same spacing and reinforcing bar properties are used at the start and end of the stem.
Rebar grade	Define the reinforcing bar grade. If you select the size from the reinforcing bar catalog, the grade value is not used.
Rebar size	Define the reinforcing bar size by selecting it from the reinforcing bar catalog.

Stirrup shape

Option	Description
	No stirrups.

Option	Description
	Open U-shape stirrup with legs.
	180-degree closed stirrups.

Stirrup dimensions

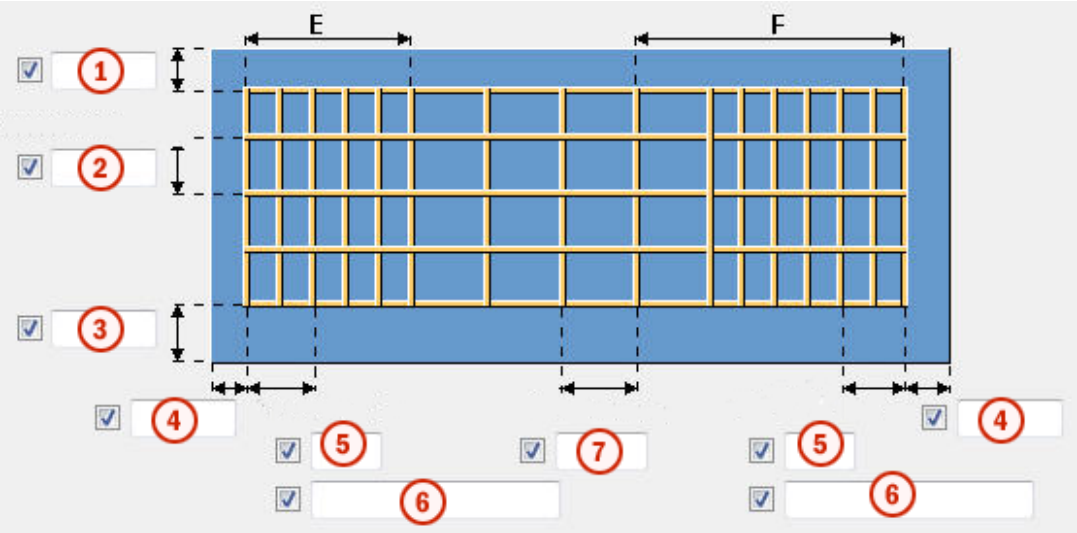


	Description
①	Define the length of the stirrup leg.
②	Define the distance from the middle of the stirrup leg to the face of the stem. To define the distance, first select **Top cover** from the **Specify** list.
③	Define the height of the stirrup. To define the height, first select **Stirrup height** from the **Specify** list.
④	Define the distance from the bottom edge of the stirrup to the bottom edge of the stem.
⑤	Define the distance from the bottom corner edge of the stirrup to the side face of the stem.

Flange tab

Use the **Flange** tab to define the reinforcing bar mesh for flanges.

Reinforcing bar mesh dimensions



	Description
1	Define the distance from the top edge of the mesh to the top edge of the flange.

	Description
2	Define the distance between two longitudinal reinforcing bars.
3	Define the distance from the bottom edge of the mesh to the bottom edge of the flange.
4	Define the distance from the side edge of the mesh to the side edge of the flange.
5	Define the number of reinforcing bars.
6	Define the reinforcing bar spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.
7	Define the maximum spacing between reinforcing bars.

Reinforcing bar properties

Define the reinforcing bar properties for both the transverse (crossing) and longitudinal reinforcing bars.

Option	Description
Size	Define the reinforcing bar size by selecting it from the reinforcing bar catalog.
Grade	Define the reinforcing bar grade. If you select the size from the reinforcing bar catalog, the grade value is not used.
Mesh catalog type	Define the mesh type by selecting if from the mesh catalog.
Ht	Define the height of the mesh.
Mesh full length	Define whether the mesh is full length or not. If you select **No**, define the end lengths for the mesh.
Left end length, E	Define the left end length of the mesh.
Right end length, F	Define the right end length of the mesh.
Min. longitudinal overlapping	Define the minimum overlap along the longitudinal direction of the mesh.
Min. cross overlapping	Define the minimum overlap along the crossing direction of the mesh.
Mesh leftover	Define whether the leftover is cut, **Cut leftover**, or not, **Overlap leftover**.

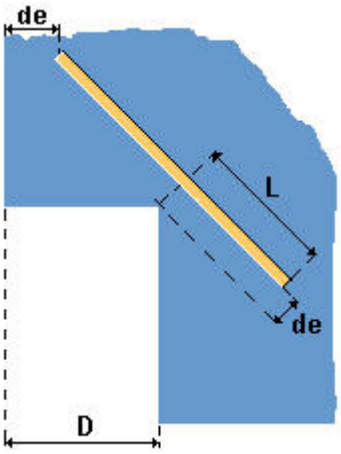
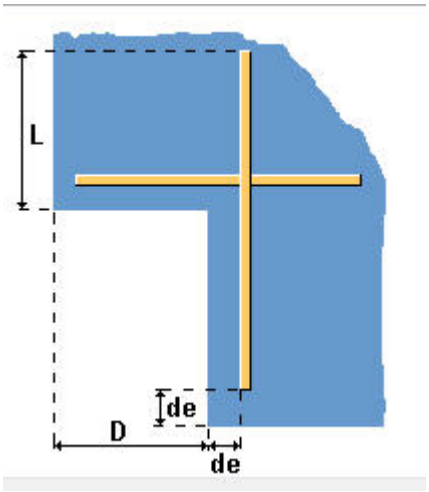
Notch/Opening tab

Use the **Notch/Opening** tab to define the reinforcement around the notches and openings.

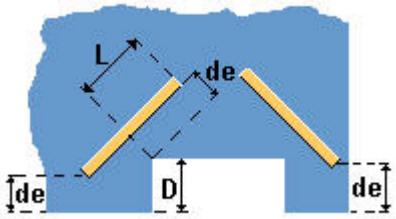
Notch/opening reinforcement properties

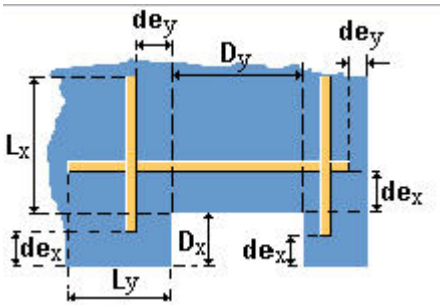
Option	Description
Min notch dimension, D	Define the minimum notch dimension.
Edge distance, de	Define the distance from the reinforcing bar to the corner of the opening.
Extension, L	Define the distance from the notch opening to the end of the reinforcing bar.
Number of bars	Define the number of reinforcing bars.
Spacing	Define the reinforcing bar spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.
Rebar size	Define the reinforcing bar size by selecting it from the reinforcing bar catalog.
Rebar grade	Define the reinforcing bar grade. If you select the size from the reinforcing bar catalog, the grade value is not used.

Corner notch reinforcement

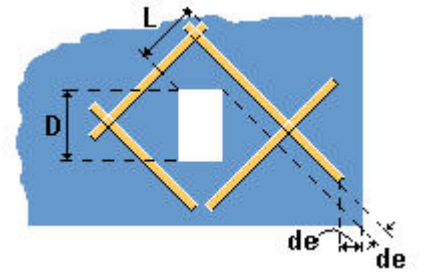
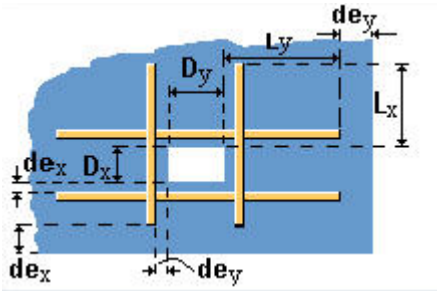
Option	Description
	Diagonal reinforcement around the corner notches.
	Parallel reinforcement around the corner notches.

Side/end notch reinforcement

Option	Description
	Diagonal reinforcement around the side or end notch.

Option	Description
	Parallel reinforcement around the side or end notch. Define both the horizontal and vertical reinforcement properties.

Opening reinforcement

Option	Description
	Diagonal reinforcement around the opening.
	Parallel reinforcement around the opening. Define both the horizontal and vertical reinforcement properties.

Edge reinforcement tab

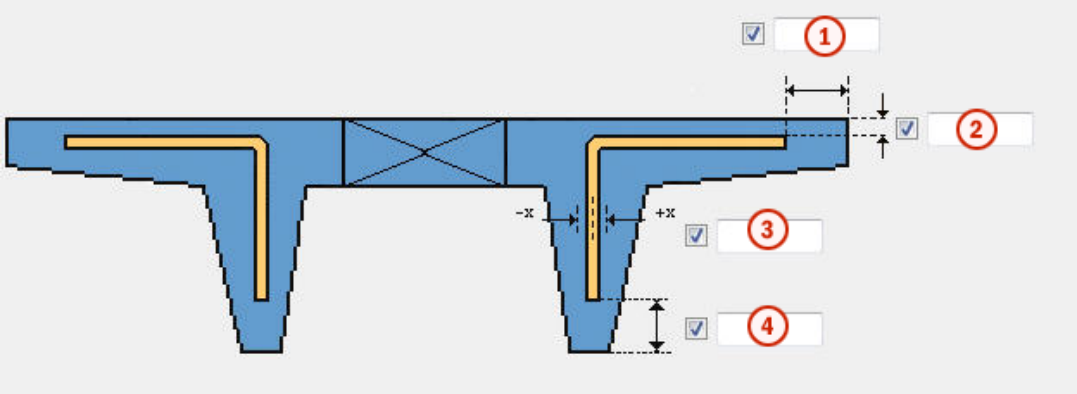
Use the **Edge reinforcement** tab to create edge reinforcement for the openings.

Reinforcement bar properties

Option	Description
Number of bars	Define the number of reinforcing bars.
Spacing	Define the reinforcement bar spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.

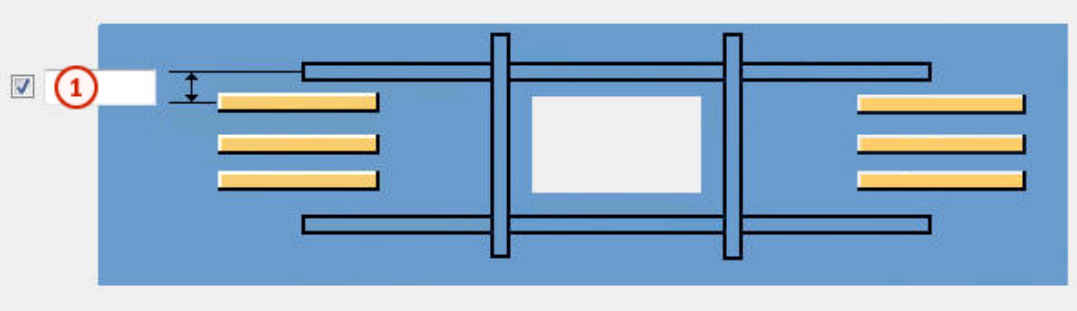
Option	Description
Rebar size	Define the reinforcing bar size by selecting it from the reinforcing bar catalog.
Rebar grade	Define the reinforcing bar grade. If you select the size from the reinforcing bar catalog, the grade value is not used.

Section view



Option	Description
1	Define the distance from the end of the reinforcing bar to the edge of the stem.
2	Define the distance from the top edge of the reinforcing bar to the face of the stem.
3	Define the distance of the reinforcing bar from the side face of the stem. Enter a positive value to move the reinforcing bar closer to the outer edge of the stem. Enter a negative value to move the reinforcing bar to the inner edge of the stem.
4	Define the distance from the bottom edge of the reinforcing bar to the bottom edge of the stem.

Plan view



Option	Description
1	Define the distance from the middle of the outmost reinforcing bar to the notch/opening reinforcement.

Attributes tab

Use the **Attributes** tab to define naming and numbering of the strands, stirrups, meshes, and bars.

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Strand code

Store strand code - With option **Store strand code** you may choose to store (or not) a user-defined textual attribute `TS_STRAND_CODE` to part.

Strand code – In field **Strand code** you may enter a string template for the value of the above mentioned user-defined attribute. The default template is the total number of created strands. The template may contain any text and the predefined fields for:

- %NUMBERS – is the numbers of created strands with different sizes and/or grades e.g. 10 (for one size & grade) or 6+4 (if two sizes and/or grades has been used).
- %SIZES – sizes of the strands e.g 1/2 or 1/2 + 3/8.
- %GRADES – grades of strands.

Strand creation at voids

Option	Description
Create strand at voids	• Select **Yes** to show the strands even though there are voids. • Select **No** to not show the strand if there is a void at any location along the strand path.
Cut strands at voids	Select whether to cut the strands at void locations. Define the cover thickness.
Min. strands length	Define the miminum strand legth.
Ignore cutouts in strand layout	Select whether cutouts are ignored when counting the strand layout.

Mesh creation at voids

Option	Description
Create meshes at voids	Select whether individual stem mesh layers are created when the layer touches a void.
Cut meshes at voids	Select whether to cut the meshes at void locations. Define the cover thickness.

Stem end reinforcement tab

Use the **Stem end reinforcement** tab to define the properties of reinforcement bars created on the stem ends.

Option	Description
Name	User-definable name of the bar. Tekla Structures uses bar names in reports and drawing lists, and to identify bars of the same type.
Grade	Steel grade of the bar.
Size	Diameter of the bar. Depending on the environment, the nominal diameter of the bar, or a mark that defines the diameter.

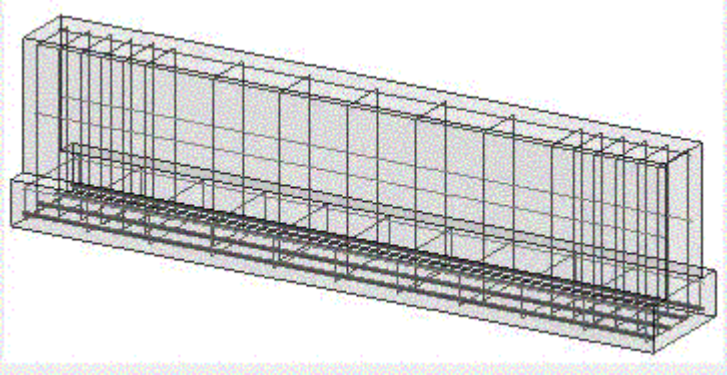
Option	Description	
Bend radius	Internal radius of the bends in the bar. You can enter a separate value for each bar bend. Separate the values with spaces. Bending radius complies with the design code you are using. Main bars, stirrups, ties, and hooks usually have their own minimum internal bending radii, which are proportional to the diameter of the reinforcing bar. The actual bending radius is normally chosen to suit the size of the mandrels on the bar-bending machine.	
Prefix	Prefix for the part position number.	
Start number	Start number for the part position number.	
Class	Use to group reinforcement. For example, you can display bars of different classes in different colors.	
Creation method	Select to create the reinforcement by **Number of bars** or **By spacings**.	

Inverted-tee beam (52)

Inverted-tee beam (52) creates typical flexural, shear, and ledge reinforcement for an inverted-tee beam.

Reinforcement created

- Prestress straight or depressed strands for flexural reinforcement in the inverted tee stem
- Mild longitudinal bars for flexural reinforcement
- Mesh for shear reinforcement in the beam stem
- Stirrups for shear reinforcement in the beam stem
- Reinforcement mesh in the inverted-tee ledges
- Stirrups for shear reinforcement in the inverted-tee ledges
- Mild reinforcement in the top of the beam that will be used later to create a composite member
- Mild reinforcement in the ends of the beam



Use for

Situation
Straight and depressed prestress strands
Varying strand pattern and strand number
To define longitudinal reinforcement in the stem
When you want reinforcement mesh in the stem
When you want stirrups in the beam stem
To use reinforcement mesh in the beam ledges
To use stirrups in the beam ledges
To define reinforcement for composite members
To add mild reinforcement to the beam ends

Before you start

- Create the concrete inverted-tee beam.
- Calculate the required area and quantity of strands and other reinforcement.
- Define other reinforcement properties, such as debonding and pull values.

Strand template tab

Use the **Strand template** tab to define the allowable grid points for strands within the beam cross section.

Grid spacing along Y and along X

You can specify the arrangement of vertical strand planes in the beam, along with the vertical distances between grid points in each plane. Strands are defined by their row and column number within the grid, but the template allows for a mixture of regular and irregular grid points with respect to both the horizontal and vertical axes.

Use the **Grid spacing along X** and **Grid spacing along Y** fields to define a list of distances between successive planes and grid points.

Separate the values with commas or spaces. The first value is the distance from the the bottom edge of the stem. Each successive value is the distance to the next grid point.

TIP Use a multiplier to define a repeating series of values. For example, 8*2 indicates eight holes with a two unit spacing.

Strand pattern tab

Use the **Strand pattern** tab to define the actual strand properties.

You place strands in the beam by defining the grid points from the strand template for each end of the strand element. You also define size, and other reinforcement properties for the strands here.

A row of data on this tab can be thought of as a group. All the strands defined in one group share the same properties and start and stop rows. A group may also refer to a list of columns in the strand template, letting you use one group definition to create a set of strands.

External file

With an external CSV file you can define an unlimited amount of individual strands.

Select to use an external file from the **Use external file** list.

Select an external CSV file by clicking **...** button.

Rebar end is same as start

Define the strand input.

- Select **Yes** to use the row and debond information of the strand start for the strands at the strand end.
- Select **No** to define an alternate row and debond information at the end of each strand.

Strand pattern definitions

Define the row and column for each end of the strand using the strand template positions in the strand grid. Strands may have different grid row positions at each end, but a strand cannot cross grid columns.

Option	Description
Strand start - Row	Define the horizontal row number in the strand template for the start of the strand.
Strand start - Debond	Define the debond length at the start of the strand.
Strand end - Row	If **Same as left** is **No**, define the horizontal row number in the strand template for the end of the strand.

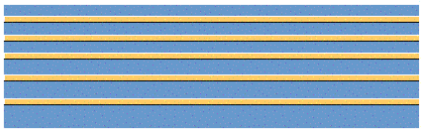
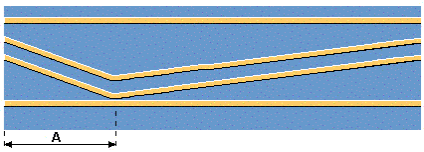
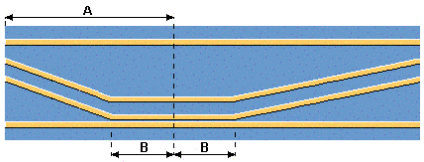
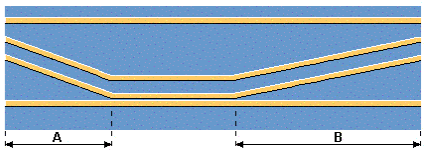
Option	Description
Strand end - Debond	If **Same as left** is **No**, define the debond length at the end of the strand.
Columns	Define a list of one or more column numbers and/or column ranges in the strand template at which a strand should be created.
Size	Define the size of the strands by selecting if from the reinforcing bar catalog.
Grade	Define the strand grade.
Pull	Define the pull force of the strands.

Strand profile tab

Use the **Strand profile** tab to define the vertical profile for strands in the beam.

Strands may be straight (though not necessarily horizontal), or depressed, using one or two control points. Note that all strands use the same general profile.

Use the following options to define the strand profile:

Option	Description
	Straight
	1-pt depressed
	2-pt depressed variable
	2-pt depressed fixed

Define the height of the strand in **Strand ht @ position**.

Depress position

Define depress positions **A** and **B** as either a fixed distance or as a percentage of the member length.

Same as A

Define whether depress position B uses the same values as depress position A in the **Two points, depressed variable** and **Two points, depressed fixed** strand profiles.

Longitudinal rebar tab

Use the **Longitudinal rebar** tab to create additional mild steel reinforcement in the beam. Reinforcement may be defined with a variety of parametric forms.

Minimum end clear distance C & D

Define the minimum end clearance distance.

Longitudinal rebar definitions

Option	Description
Grade	Define the grade of the reinforcing bar.
Size	Define the size of the reinforcing bar by selecting it from the reinforcing bar catalog.
Xloc	Specify the horizontal location of the reinforcing bar.
Yloc	Specify the vertical location of the reinforcing bar.

Length type

Define the length type for the reinforcing bar group. Enter additional information in the L, C, or D fields.

Option	Description
Full length	Reinforcing bar extends the full length of the stem, excluding the minimum end clear distance at each end. The L, C, and D fields are disabled.
Centered	Define a fixed length L for the reinforcing bar. The reinforcing bar is place at the center position in the stem. The C and D fields are disabled.

Option	Description
Known L, C	Define a fixed length L for the reinforcing bar and a distance C from the start of the strand. The D field is disabled.
Known L, C	Define a fixed length L for the reinforcing bar and a distance D from the end of the strand. The C field is disabled.
Known C, D	Define a distance C and D from the end of the strand. The L field is disabled.

Location

Specify the relative location for longitudinal bars using the following icons:

Description	Option
Top and bottom left:	
Top and bottom right:	
Top and bottom left and right:	
Top left:	
Top right:	
Top left and right:	
Bottom left:	
Bottom right:	

Description	Option
Bottom left and right:	

Stem mesh tab

Use the **Stem mesh** tab to define the mesh used in the beam stem. Mesh can run the full length of the beam, or be placed in short sections at either end of the beam.

Stem mesh type

Define the mesh type by selecting it from the **Mesh type catalog**. The mesh type also defines the mesh grade, mesh size, and bar spacing.

Stem mesh length

Specify whether the mesh is continuous over the full length of the member or is used only at the spandrel end(s) from the **Continuous** list. If **Yes** is selected from the list, the mesh extends the full length of the member. If **No** is selected from the list, mesh can be defined at each end.

Stem stirrups tab

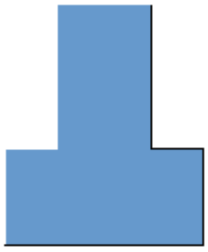
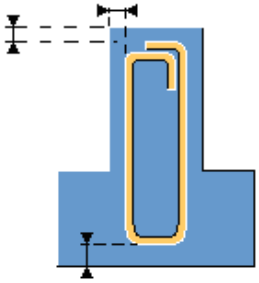
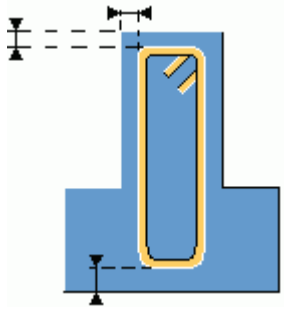
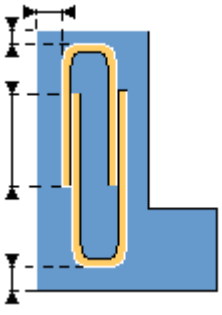
Use the **Stem stirrups** tab to define the bent bar reinforcement in the beam stem. This includes stirrup shape, bar size, end clearance, number of bars, and bar spacing in five zones along the length of the beam.

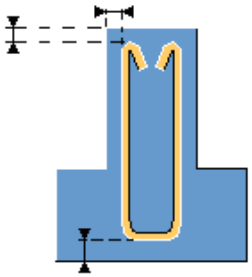
Reinforcing bar properties

Option	Description
Spacing	Define the reinforcing bar spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.
Bars	Define the number of reinforcing bars.
Rebar size	Define the reinforcing bar size by selecting it from the reinforcing bar catalog.
Same as left end	Define whether the same spacing and reinforcing bar properties are used at the right end of the stem.
Rebar grade	Define the reinforcing bar grade. If you select the size from the reinforcing bar catalog, the grade value is not used.

Stirrup shape

Use the following options to define the stem stirrups in inverted-tee beams:

Option	Description
	No stirrup
	90-degree closed stirrup
	135-degree closed stirrup
	U-Bar pairs

Option	Description
	135-degree open stirrup

Ledge mesh tab

Use the **Ledge mesh** tab to define the mesh to be used to reinforce the beam ledges. Mesh runs the full length of the beam.

Ledge mesh type

Define the mesh type by selecting it from the **Mesh type catalog**. The mesh type also defines the mesh grade, mesh size, and bar spacing.

Ledge rebar tab

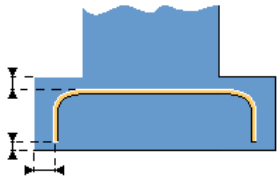
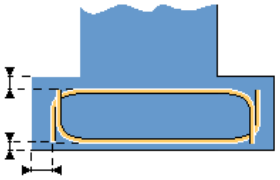
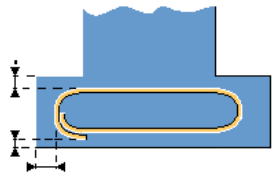
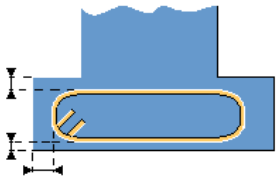
Use the **Ledge rebar** tab to define the bent bar reinforcement in the beam ledges. This includes strirrup shape, bar size, end clearance, number of bars, and bar spacing in five zones along the length of the beam.

Reinforcing bar properties

Option	Description
Spacing	Define the reinforcing bar spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.
Bars	Define the number of reinforcing bars.
Rebar size	Define the reinforcing bar size by selecting it from the reinforcing bar catalog.
Same as left end	Define whether the same spacing and reinforcing bar properties are used at the right end of the stem.
Rebar grade	Define the reinforcing bar grade. If you select the size from the reinforcing bar catalog, the grade value is not used.

Stirrup shape

Use the following options to define the ledge stirrups in inverted-tee beams:

Option	Description
	No stirrup
	Bent U-bar stirrup
	Bent U-Bar pairs
	90-degree stirrup
	135-degree stirrup

Composite tab

Use the **Composite** tab to define the bent bar reinforcement added to the top of the beam stem for subsequent creation of composite elements. Define bar size, embedment, projection, cover, end clearance, number of bars, and bar spacing in three zones along the length of the beam.

Reinforcing bar properties

Option	Description
Spacing	Define the reinforcing bar spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.
Bars	Define the number of reinforcing bars.

Option	Description
Rebar size	Define the reinforcing bar size by selecting it from the reinforcing bar catalog.
Rebar grade	Define the reinforcing bar grade. If you select the size from the reinforcing bar catalog, the grade value is not used.

End rebar tab

Use the **End rebar** tab to define the properties of bent bar reinforcement in the beam ends. Up to 6 horizontal and 6 vertical bars may be defined.

Option	Description
Size	Define the size of the reinforcing bar by selecting it from the reinforcing bar catalog.
Grade	Define the grade of the reinforcing bar.

Attributes tab

Use the **Attributes** to define naming and numbering properties of the strands, stirrups, and mesh bars.

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Strand code

Store strand code – With option **Store strand code** you may choose to store (or not) a user-defined textual attribute `TS_STRAND_CODE` to part.

Strand code – In field **Strand code** you may enter a string template for the value of the above mentioned user-defined attribute. The default template is the total number of created strands. The template may contain any text and the predefined fields for:

- %NUMBERS – is the numbers of created strands with different sizes and/or grades e.g. 10 (for one size & grade) or 6+4 (if two sizes and/or grades has been used).

- %SIZES – sizes of the strands e.g 1/2 or 1/2 + 3/8.
- %GRADES – grades of strands.

Strand creation at voids

Create strand at voids – The option **Yes** will show the strands even though the voids are present. The option **No** will not show the strand if void is present at any location along the strand path.

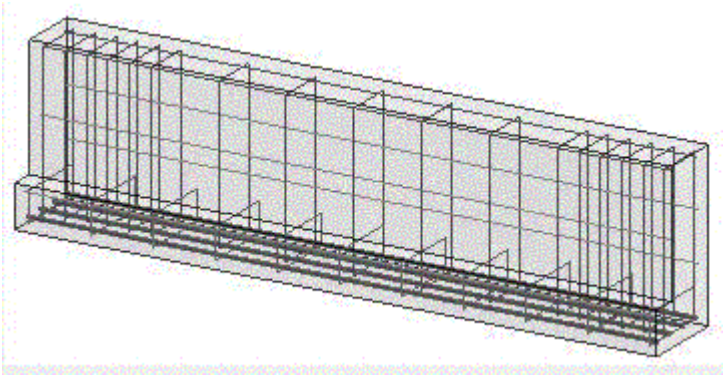
Select to cut the strands at void locations from **Cut strands at voids**.

Ledge beam (53)

Ledge beam (53) creates typical flexural, shear, and ledge reinforcement for a ledge beam.

Reinforcement created

- Prestress straight or depressed strands for flexural reinforcement in the beam stem
- Mild longitudinal bars for flexural reinforcement
- Mesh for shear reinforcement in the beam stem
- Stirrups for shear reinforcement in the beam stem
- Reinforcement mesh in the beam ledge
- Stirrups for shear reinforcement in the beam ledge
- Mild reinforcement in the top of the beam that will be used later to create a composite member
- Mild reinforcement in the ends of the beam



Use for

Situation
Straight and depressed prestress strands
Varying strand pattern and strand number

Situation
To define longitudinal reinforcement in the stem
When you want reinforcement mesh in the stem
When you want stirrups in the beam stem
To use reinforcement mesh in the beam ledge
To use stirrups in the beam ledge
To define reinforcement for composite members
To add mild reinforcement to the beam ends

Before you start

- Create the concrete ledge beam.
- Calculate the required area and quantity of strands and other reinforcement.
- Define other reinforcement properties, such as debonding and pull values.

Strand template tab

Use the **Strand template** tab to define the allowable grid points for strands within the beam cross section.

Grid spacing along Y and along X

You can specify the arrangement of vertical strand planes in the beam, along with the vertical distances between grid points in each plane. Strands are defined by their row and column number within the grid, but the template allows for a mixture of regular and irregular grid points with respect to both the horizontal and vertical axes.

Use the **Grid spacing along X** and **Grid spacing along Y** fields to define a list of distances between successive planes and grid points.

Separate the values with commas or spaces. The first value is the distance from the the bottom edge of the stem. Each successive value is the distance to the next grid point.

TIP Use a multiplier to define a repeating series of values. For example, 8*2 indicates eight holes with a two unit spacing.

Strand pattern tab

Use the **Strand pattern** tab to define the actual strand properties.

You place strands in the beam by defining the grid points from the strand template for each end of the strand element. You also define size, and other reinforcement properties for the strands here.

A row of data on this tab can be thought of as a group. All the strands defined in one group share the same properties and start and stop rows. A group may also refer to a list of columns in the strand template, letting you use one group definition to create a set of strands.

External file

With an external CSV file you can define an unlimited amount of individual strands.

Select to use an external file from the **Use external file** list.

Select an external CSV file by clicking **...** button.

Rebar end is same as start

Define the strand input.

- Select **Yes** to use the row and debond information of the strand start for the strands at the strand end.
- Select **No** to define an alternate row and debond information at the end of each strand.

Strand pattern definitions

Define the row and column for each end of the strand using the strand template positions in the strand grid. Strands may have different grid row positions at each end, but a strand cannot cross grid columns.

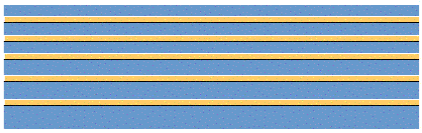
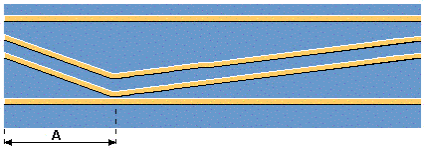
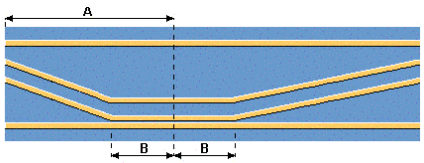
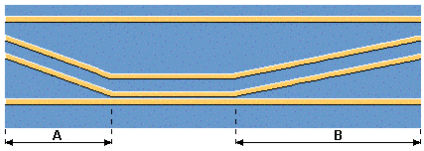
Option	Description
Strand start - Row	Define the horizontal row number in the strand template for the start of the strand.
Strand start - Debond	Define the debond length at the start of the strand.
Strand end - Row	If **Same as left** is **No**, define the horizontal row number in the strand template for the end of the strand.
Strand end - Debond	If **Same as left** is **No**, define the debond length at the end of the strand.
Columns	Define a list of one or more column numbers and/or column ranges in the strand template at which a strand should be created.
Size	Define the size of the strands by selecting if from the reinforcing bar catalog.
Grade	Define the strand grade.
Pull	Define the pull force of the strands.

Strand profile tab

Use the **Strand profile** tab to define the vertical profile for strands in the beam.

Strands may be straight (though not necessarily horizontal), or depressed, using one or two control points. Note that all strands use the same general profile.

Use the following options to define the strand profile:

Option	Description
	Straight
	1-pt depressed
	2-pt depressed variable
	2-pt depressed fixed

Define the height of the strand in **Strand ht @ position**.

Depress position

Define depress positions **A** and **B** as either a fixed distance or as a percentage of the member length.

Same as A

Define whether depress position B uses the same values as depress position A in the **Two points, depressed variable** and **Two points, depressed fixed** strand profiles.

Longitudinal rebar tab

Use the **Longitudinal rebar** tab to create additional mild steel reinforcement in the beam. Reinforcement may be defined with a variety of parametric forms.

Minimum end clear distance C & D

Define the minimum end clearance distance.

Longitudinal rebar definitions

Option	Description
Grade	Define the grade of the reinforcing bar.
Size	Define the size of the reinforcing bar by selecting it from the reinforcing bar catalog.
Xloc	Specify the horizontal location of the reinforcing bar.
Yloc	Specify the vertical location of the reinforcing bar.

Length type

Define the length type for the reinforcing bar group. Enter additional information in the L, C, or D fields.

Option	Description
Full length	Reinforcing bar extends the full length of the stem, excluding the minimum end clear distance at each end. The L, C, and D fields are disabled.
Centered	Define a fixed length L for the reinforcing bar. The reinforcing bar is place at the center position in the stem. The C and D fields are disabled.
Known L, C	Define a fixed length L for the reinforcing bar and a distance C from the start of the strand. The D field is disabled.
Known L, C	Define a fixed length L for the reinforcing bar and a distance D from the end of the strand. The C field is disabled.
Known C, D	Define a distance C and D from the end of the strand. The L field is disabled.

Location

Specify the relative location for longitudinal bars using the following icons:

Description	Option
Top and bottom left:	
Top and bottom right:	
Top and bottom left and right:	
Top left:	
Top right:	
Top left and right:	
Bottom left:	
Bottom right:	
Bottom left and right:	

Stem mesh tab

Use the **Stem mesh** tab to define the mesh used in the beam stem. Mesh can run the full length of the beam, or be placed in short sections at either end of the beam.

Stem mesh type

Define the mesh type by selecting it from the**Mesh type catalog**. The mesh type also defines the mesh grade, mesh size, and bar spacing.

Stem mesh length

Specify whether the mesh is continuous over the full length of the member or is used only at the spandrel end(s) from the **Continuous** list. If **Yes** is selected from the list, the mesh extends the full length of the member. If **No** is selected from the list, mesh can be defined at each end.

Stem stirrups tab

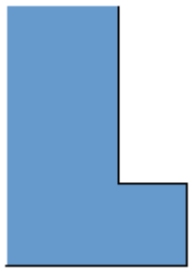
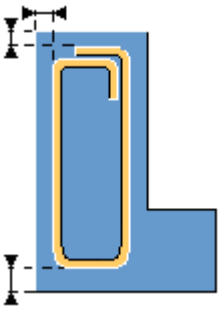
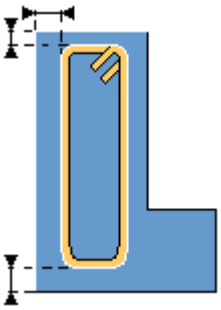
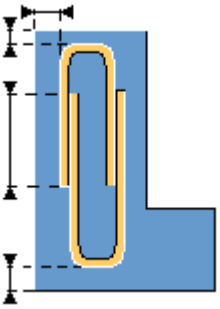
Use the **Stem stirrups** tab to define the bent bar reinforcement in the beam stem. This includes the stirrup shape, bar size, end clearance, number of bars, and bar spacing in five zones along the length of the beam.

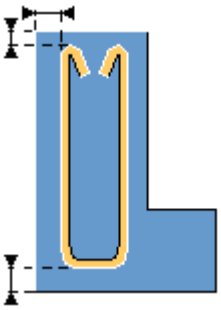
Reinforcing bar properties

Option	Description
Spacing	Define the reinforcing bar spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.
Bars	Define the number of reinforcing bars.
Rebar size	Define the reinforcing bar size by selecting it from the reinforcing bar catalog.
Same as left end	Define whether the same spacing and reinforcing bar properties are used at the right end of the stem.
Rebar grade	Define the reinforcing bar grade. If you select the size from the reinforcing bar catalog, the grade value is not used.

Stirrup shape

Use the following options to define the stem stirrups in ledge beams:

Option	Description
	No stirrup
	90-degree closed stirrup
	135-degree closed stirrup
	U-Bar pairs

Option	Description
	135-degree open stirrup

Ledge mesh tab

Use the **Ledge mesh** tab to define the mesh used to reinforce the beam ledge. Mesh runs the full length of the beam.

Ledge mesh type

Define the mesh type by selecting it from the **Mesh type catalog**. The mesh type also defines the mesh grade, mesh size, and bar spacing.

Ledge rebar tab

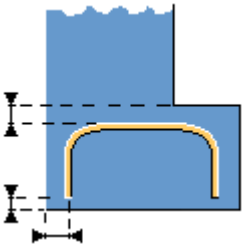
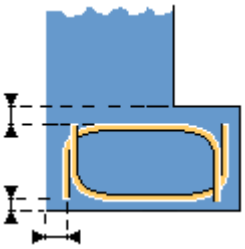
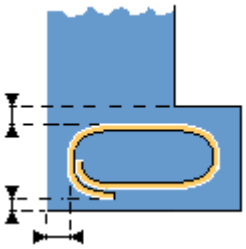
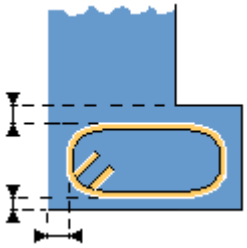
Use the **Ledge rebar** tab to define the bent bar reinforcement in the beam ledge. This includes stirrup shape, bar size, end clearance, number of bars, and bar spacing in five zones along the length of the beam.

Reinforcing bar properties

Option	Description
Spacing	Define the reinforcing bar spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.
Bars	Define the number of reinforcing bars.
Rebar size	Define the reinforcing bar size by selecting it from the reinforcing bar catalog.
Same as left end	Define whether the same spacing and reinforcing bar properties are used at the right end of the stem.
Rebar grade	Define the reinforcing bar grade. If you select the size from the reinforcing bar catalog, the grade value is not used.

Stirrup shape

Use the following options to define the ledge reinforcing bars in ledge beams:

Option	Description
	No stirrup
	Bent U-bar stirrup
	Bent U-Bar pairs
	90-degree stirrup
	135-degree stirrup

Composite tab

Use the **Composite** tab to define the bent bar reinforcement added to the top of the beam stem for subsequent creation of composite elements. Define size

and number of the bars, embedment, projection, cover, end clearance, and bar spacing in three zones along the length of the beam.

Reinforcing bar properties

Option	Description
Spacing	Define the reinforcing bar spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.
Bars	Define the number of reinforcing bars.
Rebar size	Define the reinforcing bar size by selecting it from the reinforcing bar catalog.
Rebar grade	Define the reinforcing bar grade. If you select the size from the reinforcing bar catalog, the grade value is not used.

End rebar tab

Use the **End rebar** tab to define the bent bar reinforcement in the beam ends. This includes position, orientation, bar size and length, cover, and end clearance. Up to 6 horizontal and 6 vertical bars may be defined.

Option	Description
Size	Define the size of the reinforcing bar by selecting it from the reinforcing bar catalog.
Grade	Define the grade of the reinforcing bar.

Attributes tab

Use the **Attributes** tab to define naming and numbering properties of the strands, stirrups, and mesh bars.

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Strand code

Store strand code – With option **Store strand code** you may choose to store (or not) a user-defined textual attribute `TS_STRAND_CODE` to part.

Strand code – In field **Strand code** you may enter a string template for the value of the above mentioned user-defined attribute. The default template is the total number of created strands. The template may contain any text and the predefined fields for:

- %NUMBERS – is the numbers of created strands with different sizes and/or grades e.g. 10 (for one size & grade) or 6+4 (if two sizes and/or grades has been used).
- %SIZES – sizes of the strands e.g 1/2 or 1/2 + 3/8.
- %GRADES – grades of strands.

Strand creation at voids

Create strand at voids – The option **Yes** will show the strands even though the voids are present. The option **No** will not show the strand if void is present at any location along the strand path.

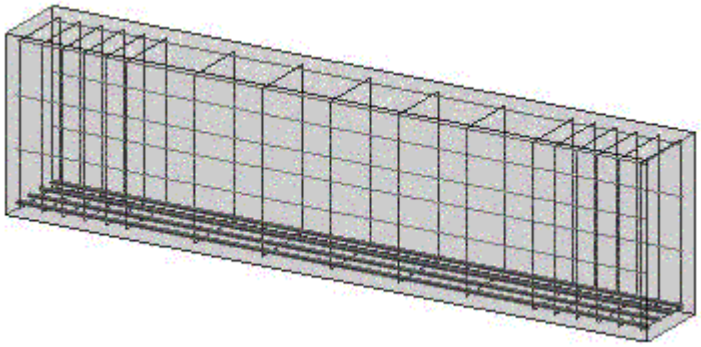
Select to cut the strands at void locations from the **Cut strands at voids** list.

Rectangular beam (54)

Rectangular beam (54) creates typical flexural and shear reinforcement for a rectangular beam.

Reinforcement created

- Prestress straight or depressed strands for flexural reinforcement in the beam
- Mild longitudinal bars for flexural reinforcement
- Mesh for shear reinforcement in the beam
- Stirrups for shear reinforcement in the beam
- Mild reinforcement in the top of the beam that will be used later to create a composite member
- Mild reinforcement in the ends of the beam



Use for

Situation
Straight and depressed prestress strands
Varying strand pattern and strand number
To define longitudinal reinforcement in the beam
When you want reinforcement mesh in the beam
When you want stirrups in the beam
To define reinforcement for composite members
To add mild reinforcement to the beam ends

Before you start

- Create the concrete rectangular beam.
- Calculate the required area and quantity of strands and other reinforcement.
- Define other reinforcement properties, such as debonding and pull values.

Strand template tab

Use the **Strand template** tab to define the allowable grid points for strands within the beam cross section.

Grid spacing along Y and along X

You can specify the arrangement of vertical strand planes in the beam, along with the vertical distances between grid points in each plane. Strands are defined by their row and column number within the grid, but the template allows for a mixture of regular and irregular grid points with respect to both the horizontal and vertical axes.

Use the **Grid spacing along X** and **Grid spacing along Y** fields to define a list of distances between successive planes and grid points.

Separate the values with commas or spaces. The first value is the distance from the the bottom edge of the stem. Each successive value is the distance to the next grid point.

TIP Use a multiplier to define a repeating series of values. For example, 8*2 indicates eight holes with a two unit spacing.

Strand pattern tab

Use the **Strand pattern** tab to define the actual strand properties.

You place strands in the beam by defining the grid points from the strand template for each end of the strand element. You also define size, and other reinforcement properties for the strands here.

A row of data on this tab can be thought of as a group. All the strands defined in one group share the same properties and start and stop rows. A group may also refer to a list of columns in the strand template, letting you use one group definition to create a set of strands.

External file

With an external CSV file you can define an unlimited amount of individual strands.

Select to use an external file from the **Use external file** list.

Select an external CSV file by clicking **...** button.

Rebar end is same as start

Define the strand input.

- Select **Yes** to use the row and debond information of the strand start for the strands at the strand end.
- Select **No** to define an alternate row and debond information at the end of each strand.

Strand pattern definitions

Define the row and column for each end of the strand using the strand template positions in the strand grid. Strands may have different grid row positions at each end, but a strand cannot cross grid columns.

Option	Description
Strand start - Row	Define the horizontal row number in the strand template for the start of the strand.
Strand start - Debond	Define the debond length at the start of the strand.
Strand end - Row	If **Same as left** is **No**, define the horizontal row number in the strand template for the end of the strand.

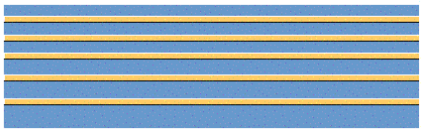
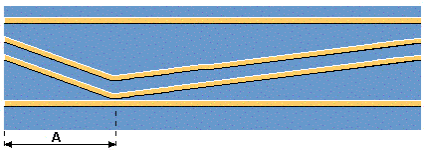
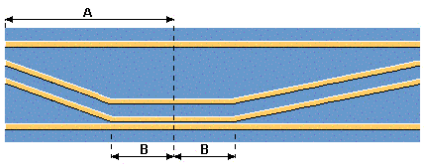
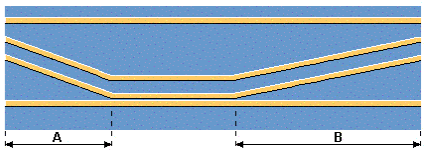
Option	Description
Strand end - Debond	If **Same as left** is **No**, define the debond length at the end of the strand.
Columns	Define a list of one or more column numbers and/or column ranges in the strand template at which a strand should be created.
Size	Define the size of the strands by selecting if from the reinforcing bar catalog.
Grade	Define the strand grade.
Pull	Define the pull force of the strands.

Strand profile tab

Use the **Strand profile** tab to define the vertical profile for strands in the beam.

Strands may be straight (though not necessarily horizontal), or depressed, using one or two control points. Note that all strands use the same general profile.

Use the following options to define the strand profile:

Option	Description
	Straight
	1-pt depressed
	2-pt depressed variable
	2-pt depressed fixed

Define the height of the strand in **Strand ht @ position**.

Depress position

Define depress positions **A** and **B** as either a fixed distance or as a percentage of the member length.

Same as A

Define whether depress position B uses the same values as depress position A in the **Two points, depressed variable** and **Two points, depressed fixed** strand profiles.

Longitudinal rebar tab

Use the **Longitudinal rebar** tab to define the additional mild steel reinforcement in the beam. Reinforcement may be defined with a variety of parametric forms.

Minimum end clear distance C & D

Define the minimum end clearance distance.

Longitudinal rebar definitions

Option	Description
Grade	Define the grade of the reinforcing bar.
Size	Define the size of the reinforcing bar by selecting it from the reinforcing bar catalog.
Xloc	Specify the horizontal location of the reinforcing bar.
Yloc	Specify the vertical location of the reinforcing bar.

Length type

Define the length type for the reinforcing bar group. Enter additional information in the L, C, or D fields.

Option	Description
Full length	Reinforcing bar extends the full length of the stem, excluding the minimum end clear distance at each end. The L, C, and D fields are disabled.
Centered	Define a fixed length L for the reinforcing bar. The reinforcing bar is place at the center position in the stem. The C and D fields are disabled.

Option	Description
Known L, C	Define a fixed length L for the reinforcing bar and a distance C from the start of the strand. The D field is disabled.
Known L, C	Define a fixed length L for the reinforcing bar and a distance D from the end of the strand. The C field is disabled.
Known C, D	Define a distance C and D from the end of the strand. The L field is disabled.

Location

Specify the relative location for longitudinal bars using the following icons:

Description	Option
Top and bottom left:	
Top and bottom right:	
Top and bottom left and right:	
Top left:	
Top right:	
Top left and right:	
Bottom left:	
Bottom right:	

Description	Option
Bottom left and right:	

Stem mesh tab

Use the **Stem mesh** tab to define the mesh used in the beam. Mesh can run the full length of the beam, or be placed in short sections at either end of the beam.

Stem mesh type

Define the mesh type by selecting it from the **Mesh type catalog**. The mesh type also defines the mesh grade, mesh size, and bar spacing.

Stem mesh length

Specify whether the mesh is continuous over the full length of the member or is used only at the spandrel end(s) from the **Continuous** list. If **Yes** is selected from the list, the mesh extends the full length of the member. If **No** is selected from the list, mesh can be defined at each end.

Stem stirrups tab

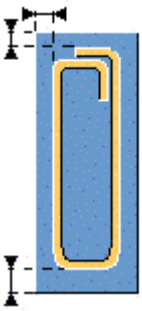
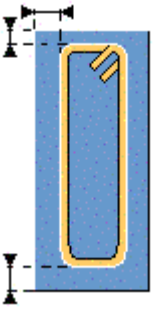
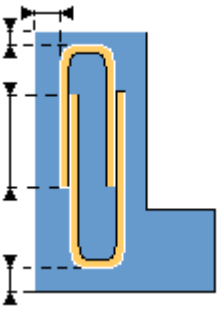
Use the **Stem stirrups** tab to define the bent bar reinforcement in the beam. This includes stirrup shape, bar properties, end clearance, and bar spacing in five zones along the length of the beam.

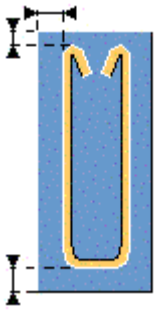
Reinforcing bar properties

Option	Description
Spacing	Define the reinforcing bar spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.
Bars	Define the number of reinforcing bars.
Rebar size	Define the reinforcing bar size by selecting it from the reinforcing bar catalog.
Same as left end	Define whether the same spacing and reinforcing bar properties are used at the right end of the stem.
Rebar grade	Define the reinforcing bar grade. If you select the size from the reinforcing bar catalog, the grade value is not used.

Stirrup shape

Use the following options to define the stem stirrups in ledge beams:

Option	Description
	No stirrup
	90-degree closed stirrup
	135-degree closed stirrup
	U-Bar pairs

Option	Description
	135-degree open stirrup

Composite tab

Use the **Composite** tab to define the bent bar reinforcement added to the top of the beam for subsequent creation of composite elements. Define bar size, embedment, projection, cover, end clearance, number of bars, and bar spacing in three zones along the length of the beam.

Reinforcing bar properties

Option	Description
Spacing	Define the reinforcing bar spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.
Bars	Define the number of reinforcing bars.
Rebar size	Define the reinforcing bar size by selecting it from the reinforcing bar catalog.
Rebar grade	Define the reinforcing bar grade. If you select the size from the reinforcing bar catalog, the grade value is not used.

End rebar tab

Use the **End rebar** tab to define the bent bar reinforcement in the beam ends. This includes position, orientation, bar size and length, cover and end clearance. Up to 6 horizontal and 6 vertical bars may be defined.

Option	Description
Size	Define the size of the reinforcing bar by selecting it from the reinforcing bar catalog.
Grade	Define the grade of the reinforcing bar.

Attributes tab

Use the **Attributes** to define naming and numbering properties of the strands, stirrups, and mesh bars.

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Strand code

Store strand code – With option **Store strand code** you may choose to store (or not) a user-defined textual attribute `TS_STRAND_CODE` to part.

Strand code – In field **Strand code** you may enter a string template for the value of the above mentioned user-defined attribute. The default template is the total number of created strands. The template may contain any text and the predefined fields for:

- %NUMBERS – is the numbers of created strands with different sizes and/or grades e.g. 10 (for one size & grade) or 6+4 (if two sizes and/or grades has been used).
- %SIZES – sizes of the strands e.g 1/2 or 1/2 + 3/8.
- %GRADES – grades of strands.

Strand creation at voids

Create strand at voids – The option **Yes** will show the strands even though the voids are present. The option **No** will not show the strand if void is present at any location along the strand path.

Select to cut the strands at void locations from **Cut strands at voids**.

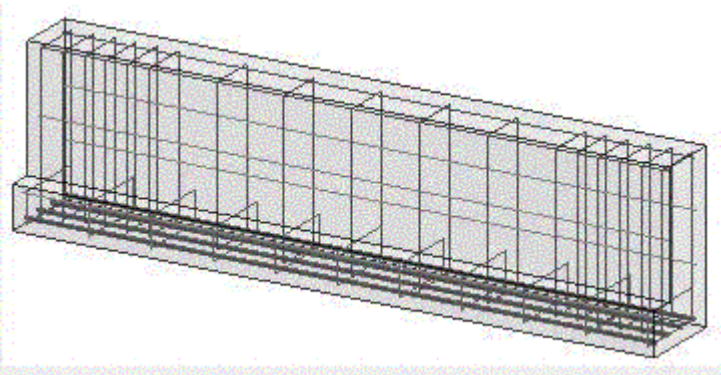
Ledge spandrel (55)

Ledge spandrel (55) creates typical flexural, shear, and ledge reinforcement for a ledge spandrel.

Reinforcement created

- Prestress straight or depressed strands for flexural reinforcement in the spandrel stem

- Mild longitudinal bars for flexural reinforcement
- Mesh for shear reinforcement in the spandrel stem
- Stirrups for shear reinforcement in the spandrel stem
- Reinforcement mesh in the spandrel ledge
- Stirrups for shear reinforcement in the spandrel ledge
- Mild reinforcement in the ends of the spandrel



Use for

Situation
Straight and depressed prestress strands
Varying strand pattern and strand number
To define longitudinal reinforcement in the stem
When you want reinforcement mesh in the stem
When you want stirrups in the spandrel stem
To use reinforcement mesh in the spandrel ledge
To use stirrups in the spandrel ledge
To add mild reinforcement to the spandrel ends

Before you start

- Create the concrete ledge spandrel.
- Calculate the required area and quantity of strands and other reinforcement.
- Define other reinforcement properties, such as debonding and pull values.

Strand template tab

Use the **Strand template** tab to define the allowable grid points for strands within the spandrel cross section.

Grid spacing along Y and along X

You can specify the arrangement of vertical strand planes in the spandrel, along with the vertical distances between grid points in each plane. Strands are defined by their row and column number within the grid, but the template allows for a mixture of regular and irregular grid points with respect to both the horizontal and vertical axes.

Use the **Grid spacing along X** and **Grid spacing along Y** fields to define a list of distances between successive planes and grid points.

Separate the values with commas or spaces. The first value is the distance from the the bottom edge of the stem. Each successive value is the distance to the next grid point.

TIP Use a multiplier to define a repeating series of values. For example, 8*2 indicates eight holes with a two unit spacing.

Strand pattern tab

Use the **Strand pattern** tab to define the actual strand properties.

You place strands in the spandrel by defining the grid points from the strand template for each end of the strand element. You also define size, and other reinforcement properties for the strands here.

A row of data on this tab can be thought of as a group. All the strands defined in one group share the same properties and start and stop rows. A group may also refer to a list of columns in the strand template, letting you use one group definition to create a set of strands.

External file

With an external CSV file you can define an unlimited amount of individual strands.

Select to use an external file from the **Use external file** list.

Select an external CSV file by clicking **...** button.

Rebar end is same as start

Define the strand input.

- Select **Yes** to use the row and debond information of the strand start for the strands at the strand end.
- Select **No** to define an alternate row and debond information at the end of each strand.

Strand pattern definitions

Define the row and column for each end of the strand using the strand template positions in the strand grid. Strands may have different grid row positions at each end, but a strand cannot cross grid columns.

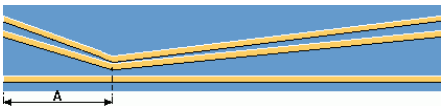
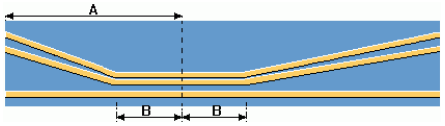
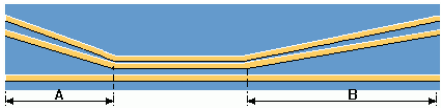
Option	Description
Strand start - Row	Define the horizontal row number in the strand template for the start of the strand.
Strand start - Debond	Define the debond length at the start of the strand.
Strand end - Row	If **Same as left** is **No**, define the horizontal row number in the strand template for the end of the strand.
Strand end - Debond	If **Same as left** is **No**, define the debond length at the end of the strand.
Columns	Define a list of one or more column numbers and/or column ranges in the strand template at which a strand should be created.
Size	Define the size of the strands by selecting if from the reinforcing bar catalog.
Grade	Define the strand grade.
Pull	Define the pull force of the strands.

Strand profile tab

Use the **Strand profile** tab to define the vertical profile for strands in the spandrel.

Strands may be straight (though not necessarily horizontal), or depressed, using one or two control points. Note that all strands use the same general profile.

Use the following options to define the strand profile:

Option	Description
	Straight
	1-pt depressed
	2-pt depressed variable
	2-pt depressed fixed

Define the height of the strand in **Strand ht @ position**.

Depress position

Define depress positions **A** and **B** as either a fixed distance or as a percentage of the member length.

Same as A

Define whether depress position B uses the same values as depress position A in the **Two points, depressed variable** and **Two points, depressed fixed** strand profiles.

Longitudinal rebar tab

Use the **Longitudinal rebar** tab to define the additional mild steel reinforcement in the spandrel. Reinforcement may be defined with a variety of parametric forms.

External file

Select to use an external file from the **Use external file** list.

Select an external CSV file by clicking **...** button.

Minimum end clear distance C & D

Define the minimum end clearance distance.

Longitudinal rebar definitions

Option	Description
Grade	Define the grade of the reinforcing bar.
Size	Define the size of the reinforcing bar by selecting it from the reinforcing bar catalog.
Xloc	Specify the horizontal location of the reinforcing bar.
Yloc	Specify the vertical location of the reinforcing bar.

Length type

Define the length type for the reinforcing bar group. Enter additional information in the L, C, or D fields.

Option	Description
Full length	Reinforcing bar extends the full length of the stem, excluding the minimum end clear distance at each end. The L, C, and D fields are disabled.

Option	Description
Centered	Define a fixed length L for the reinforcing bar. The reinforcing bar is place at the center position in the stem. The C and D fields are disabled.
Known L, C	Define a fixed length L for the reinforcing bar and a distance C from the start of the strand. The D field is disabled.
Known L, C	Define a fixed length L for the reinforcing bar and a distance D from the end of the strand. The C field is disabled.
Known C, D	Define a distance C and D from the end of the strand. The L field is disabled.

Location

Specify the relative location for longitudinal bars using the following icons:

Description	Option
Top and bottom left:	
Top and bottom right:	
Top and bottom left and right:	
Top left:	
Top right:	
Top left and right:	

Description	Option
Bottom left:	
Bottom right:	
Bottom left and right:	

Stem mesh tab

Use the **Stem mesh** tab to define the mesh to be used in the spandrel stem. Mesh can run the full length of the spandrel, or be placed in short sections at either end.

Stem mesh type

Define the mesh type by selecting it from the **Mesh type catalog**. The mesh type also defines the mesh grade, mesh size, and bar spacing.

Stem mesh length

Specify whether the mesh is continuous over the full length of the member or is used only at the spandrel end(s) from the **Continuous** list. If **Yes** is selected from the list, the mesh extends the full length of the member. If **No** is selected from the list, mesh can be defined at each end.

Stem stirrups tab

Use the **Stem stirrups** tab to define the bent bar reinforcement in the spandrel stem. This includes stirrup shape, bar size, end clearance, number of bars, and bar spacing in eight zones along the length of the spandrel.

External file

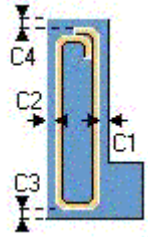
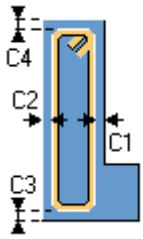
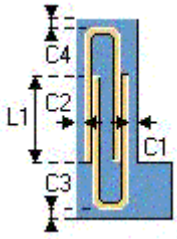
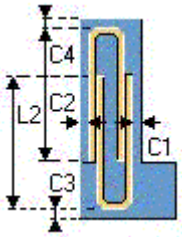
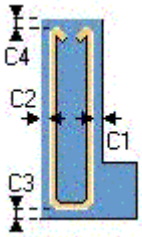
Select to use an external file from the **Use external file** list.

Select an external CSV file by clicking **...** button.

Stirrup type

Stirrups are defined in groups, and a row of data on this tab represents the definition for a group. All the stirrups defined in one group share the same properties, including stirrup type. Clear cover definitions apply to all groups.

The following stirrup types may be used in ledge spandrels:

Option	Description
	90-degree closed stirrup
	135-degree closed stirrup
	Lapped pair
	Fixed pair
	Open U

If you select **Closed** or **Follow borders**, you can define advanced hook properties.

Hook

You can select to create a hook or a custom hook from the **Hook type** list.

If you create a hook, you can define hook **Angle**, **Radius**, and **Length**.

Ledge mesh tab

Use the **Ledge mesh** tab to define the mesh used to reinforce the spandrel ledge. Mesh runs the full length of the spandrel.

Ledge mesh type

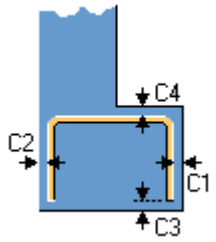
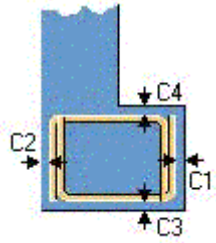
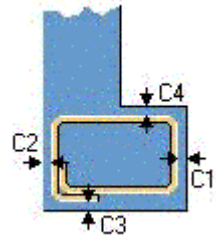
Define the mesh type by selecting it from the **Mesh type catalog**. The mesh type also defines the mesh grade, mesh size, and bar spacing.

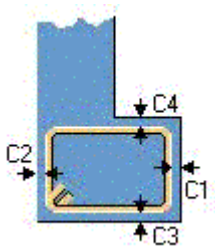
Ledge rebar tab

Use the **Ledge rebar** tab to define the bent bar reinforcement in the spandrel ledges. This includes strirrup shape, bar size, end clearance, number of bars, and bar spacing in eight zones along the length of the spandrel.

Ledge stirrups

Ledge stirrups are defined in groups, using the same concept as stem stirrups. The following ledge stirrup types may be used:

Option	Description
	Bent U-bar stirrup
	Bent U-Bar pairs
	90-degree stirrup

Option	Description
	135-degree stirrup Closed

If you select **Closed**, you can define advanced hook properties.

Hook

You can select to create a hook or a custom hook from the **Hook type** list.

If you create a hook, you can define hook **Angle**, **Radius**, and **Length**.

End rebar tab

Use the **End rebar** tab to define the bent bar reinforcement and the secondary rebar set in the spandrel ends. This includes position, orientation, bar size and length, cover, and end clearance. Up to 6 horizontal bars may be defined.

Option	Description
Size	Define the size of the reinforcing bar by selecting it from the reinforcing bar catalog.
Grade	Define the grade of the reinforcing bar.

Attributes tab

Use the **Attributes** to define naming and numbering properties of the strands, stirrups, and bars.

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Strand code

Store strand code – With option **Store strand code** you may choose to store (or not) a user-defined textual attribute `TS_STRAND_CODE` to part.

Strand code – In field **Strand code** you may enter a string template for the value of the above mentioned user-defined attribute. The default template is the total number of created strands. The template may contain any text and the predefined fields for:

- %NUMBERS – is the numbers of created strands with different sizes and/or grades e.g. 10 (for one size & grade) or 6+4 (if two sizes and/or grades has been used).
- %SIZES – sizes of the strands e.g 1/2 or 1/2 + 3/8.
- %GRADES – grades of strands.

Strand creation at voids

Create strand at voids – The option **Yes** will show the strands even though the voids are present. The option **No** will not show the strand if void is present at any location along the strand path.

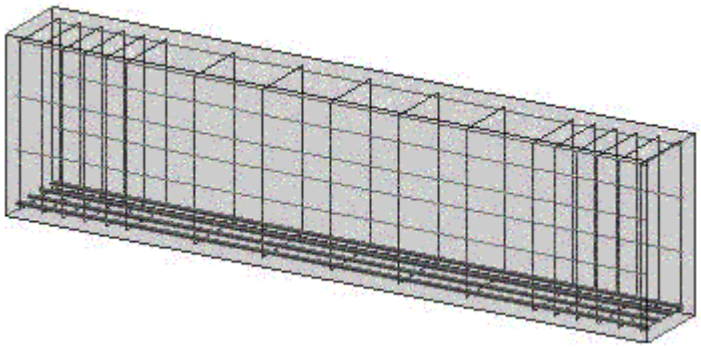
Select to cut the strands at void locations from **Cut strands at voids**.

Rectangular spandrel (56)

Rectangular spandrel (56) creates typical flexural and shear reinforcement for a rectangular spandrel.

Reinforcement created

- Prestress straight or depressed strands for flexural reinforcement in the spandrel
- Mild longitudinal bars for flexural reinforcement
- Mesh for shear reinforcement in the spandrel
- Stirrups for shear reinforcement in the spandrel
- Mild reinforcement in the ends of the spandrel



Use for

Situation	More information
Straight and depressed prestress strands	
Varying strand pattern and strand number	
To define longitudinal reinforcement in the spandrel	
When you want reinforcement mesh in the spandrel	
When you want stirrups in the spandrel	
To add mild reinforcement to the spandrel ends	

Before you start

- Create the concrete rectangular spandrel.
- Calculate the required area and quantity of strands and other reinforcement.
- Define other reinforcement properties, such as debonding and pull values.

Strand template tab

Use the **Stran template** tab to define the allowable grid points for strands within the spandrel cross section.

Grid spacing along Y and along X

You can specify the arrangement of vertical strand planes in the beam, along with the vertical distances between grid points in each plane. Strands are defined by their row and column number within the grid, but the template allows for a mixture of regular and irregular grid points with respect to both the horizontal and vertical axes.

Use the **Grid spacing along X** and **Grid spacing along Y** fields to define a list of distances between successive planes and grid points.

Separate the values with commas or spaces. The first value is the distance from the the bottom edge of the stem. Each successive value is the distance to the next grid point.

TIP Use a multiplier to define a repeating series of values. For example, 8*2 indicates eight holes with a two unit spacing.

Strand pattern tab

Use the **Strand pattern** tab to define the actual strand properties.

You place strands in the spandrel by defining the grid points from the strand template for each end of the strand element. You also define size, and other reinforcement properties for the strands here.

A row of data on this tab can be thought of as a group. All the strands defined in one group share the same properties and start and stop rows. A group may also refer to a list of columns in the strand template, letting you use one group definition to create a set of strands.

External file

With an external CSV file you can define an unlimited amount of individual strands.

Select to use an external file from the **Use external file** list.

Select an external CSV file by clicking **...** button.

Rebar end is same as start

Define the strand input.

- Select **Yes** to use the row and debond information of the strand start for the strands at the strand end.
- Select **No** to define an alternate row and debond information at the end of each strand.

Strand pattern definitions

Define the row and column for each end of the strand using the strand template positions in the strand grid. Strands may have different grid row positions at each end, but a strand cannot cross grid columns.

Option	Description
Strand start - Row	Define the horizontal row number in the strand template for the start of the strand.
Strand start - Debond	Define the debond length at the start of the strand.
Strand end - Row	If **Same as left** is **No**, define the horizontal row number in the strand template for the end of the strand.
Strand end - Debond	If **Same as left** is **No**, define the debond length at the end of the strand.
Columns	Define a list of one or more column numbers and/or column ranges in the strand template at which a strand should be created.
Size	Define the size of the strands by selecting if from the reinforcing bar catalog.

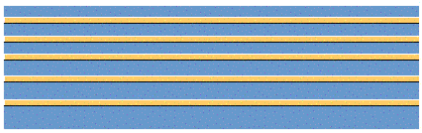
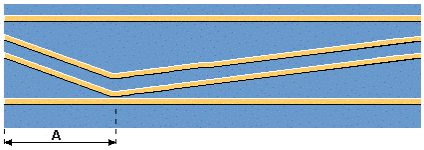
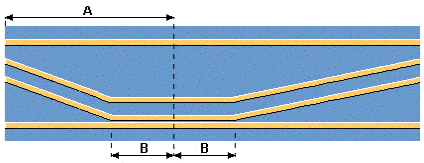
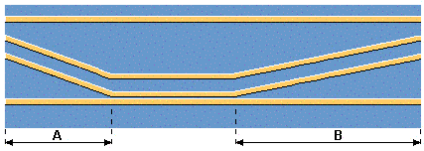
Option	Description
Grade	Define the strand grade.
Pull	Define the pull force of the strands.

Strand profile tab

Use the **Strand profile** tab to define the vertical profile for strands in the spandrel.

Strands may be straight (though not necessarily horizontal), or depressed, using one or two control points. Note that all strands use the same general profile.

Use the following options to define the strand profile:

Option	Description
	Straight
	1-pt depressed
	2-pt depressed variable
	2-pt depressed fixed

Define the height of the strand in **Strand ht @ position**.

Depress position

Define depress positions **A** and **B** as either a fixed distance or as a percentage of the member length.

NOTE If no specific value is given the “Ht” shall be calculated as Minimum. So the strand will be straight unless it needs to be depressed because of the above strand going below the straight position.

Same as A

Define whether depress position B uses the same values as depress position A in the **Two points, depressed variable** and **Two points, depressed fixed** strand profiles.

Longitudinal rebar tab

Use the **Longitudinal rebar** tab to define the additional mild steel reinforcement in the spandrel. Reinforcement may be defined with a variety of parametric forms.

Minimum end clear distance C & D

Define the minimum end clearance distance.

Longitudinal rebar definitions

Option	Description
Grade	Define the grade of the reinforcing bar.
Size	Define the size of the reinforcing bar by selecting it from the reinforcing bar catalog.
Xloc	Specify the horizontal location of the reinforcing bar.
Yloc	Specify the vertical location of the reinforcing bar.

Length type

Define the length type for the reinforcing bar group. Enter additional information in the L, C, or D fields.

Option	Description
Full length	Reinforcing bar extends the full length of the stem, excluding the minimum end clear distance at each end. The L, C, and D fields are disabled.
Centered	Define a fixed length L for the reinforcing bar. The reinforcing bar is place at the center position in the stem. The C and D fields are disabled.
Known L, C	Define a fixed length L for the reinforcing bar and a distance C from the start of the strand. The D field is disabled.
Known L, C	Define a fixed length L for the reinforcing bar and a distance D from the end of the strand. The C field is disabled.

Option	Description
Known C, D	Define a distance C and D from the end of the strand. The L field is disabled.

Location

Specify the relative location for longitudinal bars using the following icons:

Description	Option
Top and bottom left:	
Top and bottom right:	
Top and bottom left and right:	
Top left:	
Top right:	
Top left and right:	
Bottom left:	
Bottom right:	
Bottom left and right:	

Stem mesh tab

Use the **Stem mesh** tab to define the mesh used in the spandrel. Mesh can run the full length of the spandrel, or be placed in short sections at either end.

Stem mesh type

Define the mesh type by selecting it from the **Mesh type catalog**. The mesh type also defines the mesh grade, mesh size, and bar spacing.

Stem mesh length

Specify whether the mesh is continuous over the full length of the member or is used only at the spandrel end(s) from the **Continuous** list. If **Yes** is selected from the list, the mesh extends the full length of the member. If **No** is selected from the list, mesh can be defined at each end.

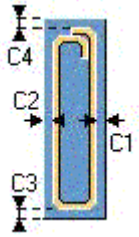
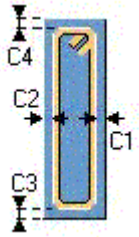
Stem stirrups tab

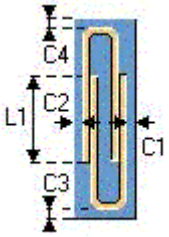
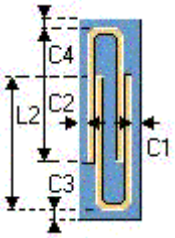
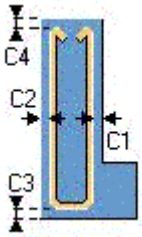
Use the **Stem stirrups** tab to define the bent bar reinforcement in the spandrel. This includes stirrup shape, bar size, end clearance, number of bars, and bar spacing in eight zones along the length of the spandrel.

Stirrup type

Stirrups are defined in groups, and a row of data on this tab represents the definition for a group. All the stirrups defined in one group share the same properties, including stirrup type. Clear cover definitions apply to all groups.

Use the following options to define the stem stirrups in rectangular spandrels:

Option	Description
	90-degree closed stirrup
	135-degree closed stirrup

Option	Description
	U-Bar lapped pairs
	U-Bar fixed pairs
	Open U stirrup

If you select **Closed**, you can define advanced hook properties.

Hook

You can select to create a hook or a custom hook from **Hook type** dropdown list.

If you create a hook, you can define hook **Angle**, **Radius**, and **Length**.

End rebar tab

Use the **End rebar** tab to define the bent bar reinforcement in the spandrel ends. This includes position, orientation, bar size and length, cover, and end clearance. Up to 6 horizontal bars may be defined.

Option	Description
Size	Define the size of the reinforcing bar by selecting it from the reinforcing bar catalog.
Grade	Define the grade of the reinforcing bar.

Attributes tab

Use the **Attributes** to define naming and numbering properties of the strands, stirrups, and mesh bars.

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Strand code

Store strand code – With option **Store strand code** you may choose to store (or not) a user-defined textual attribute `TS_STRAND_CODE` to part.

Strand code – In field **Strand code** you may enter a string template for the value of the above mentioned user-defined attribute. The default template is the total number of created strands. The template may contain any text and the predefined fields for:

- %NUMBERS – is the numbers of created strands with different sizes and/or grades e.g. 10 (for one size & grade) or 6+4 (if two sizes and/or grades has been used).
- %SIZES – sizes of the strands e.g 1/2 or 1/2 + 3/8.
- %GRADES – grades of strands.

Strand creation at voids

Create strand at voids – The option **Yes** will show the strands even though the voids are present. The option **No** will not show the strand if void is present at any location along the strand path.

Select to cut the strands at void locations from **Cut strands at voids**.

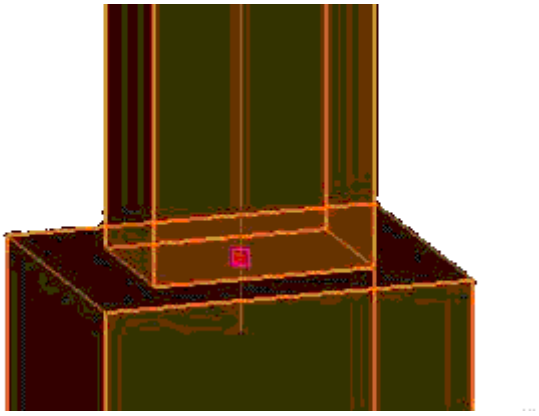
Columns (57)

Columns (57) creates typical longitudinal and tie reinforcement for a column.

Reinforcement created

- Prestress strands for longitudinal reinforcement in a column
- Mild longitudinal bars for longitudinal reinforcement in a column

- Reinforcement for composite rectangular columns



Use for

Situation	More information
To define longitudinal prestress reinforcement size and location	
To define longitudinal mild reinforcement size and extent	
To define tie size and spacings	

Before you start

- Create the column.
- Calculate the required area and quantity of strands and other reinforcement.
- Define other reinforcement properties.

Strand reinforcement tab

Use the **Strand reinforcement** tab to define strand attributes and for a description of the geometric layout of pre-stress strands in the column.

This tab lets you define the strand reinforcement in the primary section of the column. Up to 8 strands may be defined.

Option	Description
Size	Define the size of the strands by selecting if from the reinforcing bar catalog.
Grade	Define the strand grade.

Option	Description
Pull	Define the pull force of the strands.

Specify strand-centerline-to-surface dimension, **de**. For corner strands 1-4, this distance defines the strand centerline position relative to both adjacent column faces. For strands 5-8, the strand is located on the face centerline, offset from the face by distance **de**.

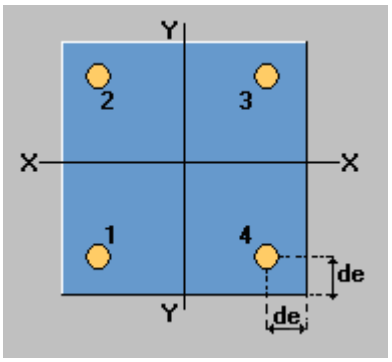
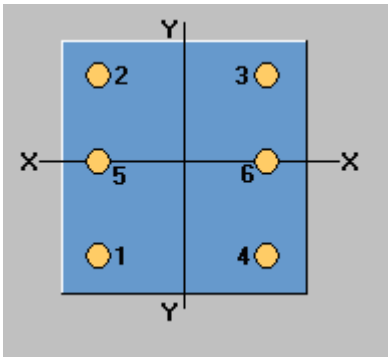
Longitudinal rebar tab

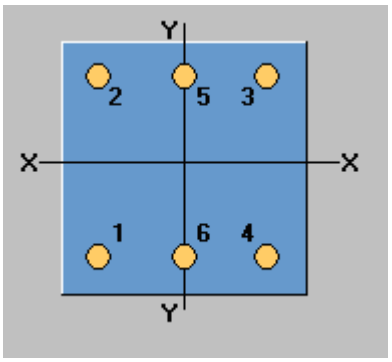
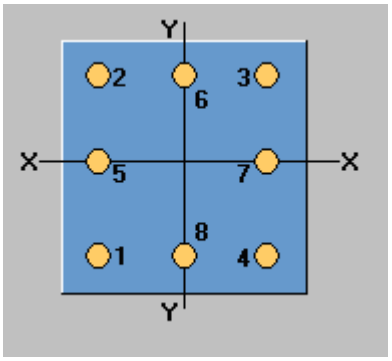
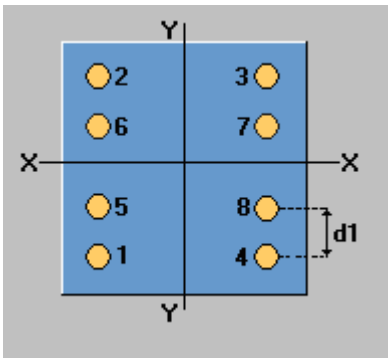
Use the **Longitudinal rebar** tab to define the longitudinal reinforcement to be used in the primary section of the column.

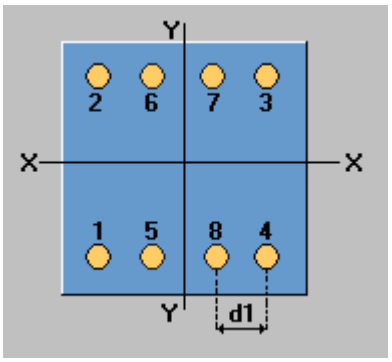
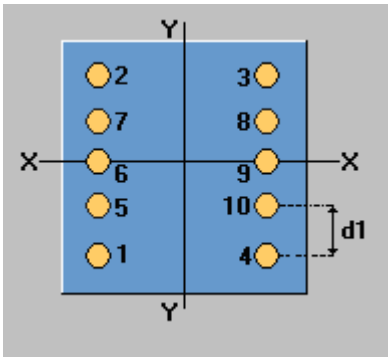
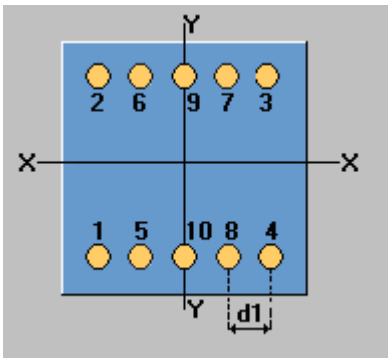
Reinforcement pattern

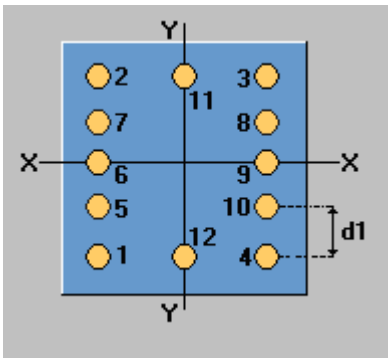
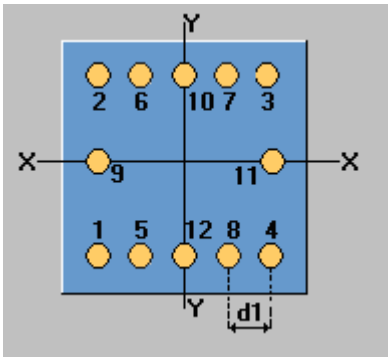
Longitudinal bars are presumed to run the entire length of the column (less the End Clearance distance at each end of the member). Standard longitudinal reinforcement configurations are provided in a convenient drop-down, and up to 12 bars may be defined.

Select the desired reinforcement pattern from the drop-down combo-box. There are 10 predefined patterns:

Description	Option
4 Bar Symmetrical	
6 Bar X-X	

Description	Option
6 Bar Y-Y	
8 Bar Symmetrical	
8 Bar X-X	

Description	Option
8 Bar Y-Y	
10 Bar X-X	
10 Bar Y-Y	

Description	Option
12 Bar X-X	
12 Bar Y-Y	

Rebar properties

Option	Description
de	Specify the reinforcing bar-centerline-to-surface dimension. This value is used to locate most of the bars in each pattern.
d1	Specify the dimension, which is used in only the 10- and 12-bar patterns to locate off-axis bars. This value should default to the half the appropriate primary section dimension less **de**, divided by 2.
End clear distance	Specify the end clear distance for both ends of the longitudinal reinforcing bars.
Size	Select the size of the bar at each bar location.
Grade	Select the grade of bar at each bar location.

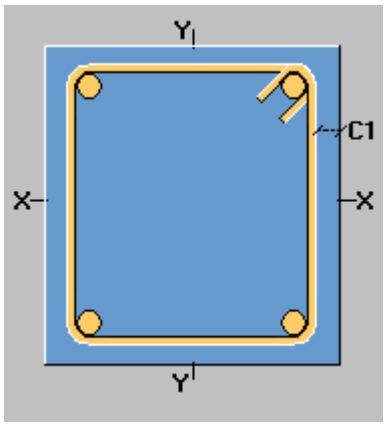
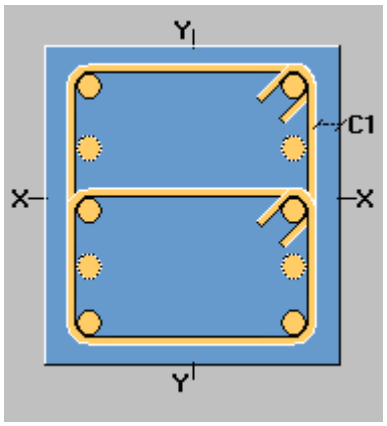
Primary ties tab

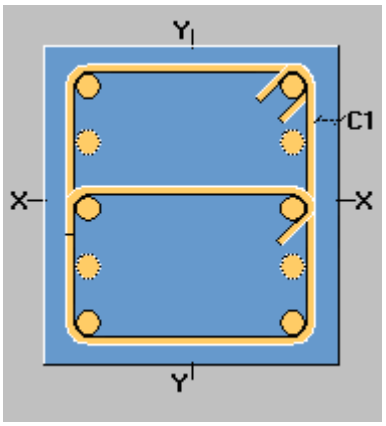
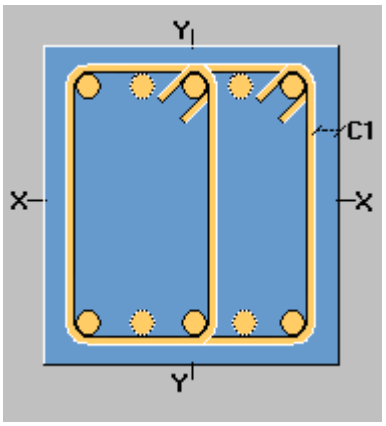
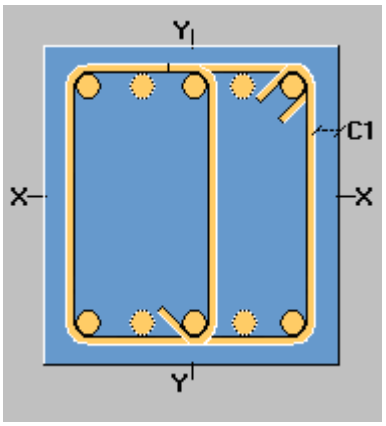
Use the **Primary ties** tab to define the ties used in the primary section of the column.

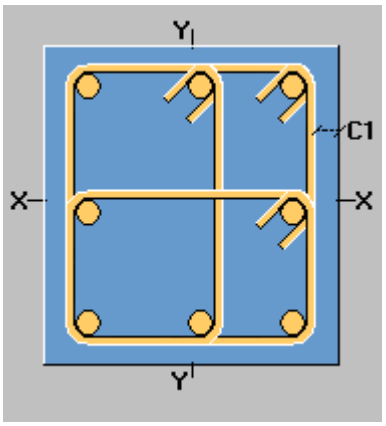
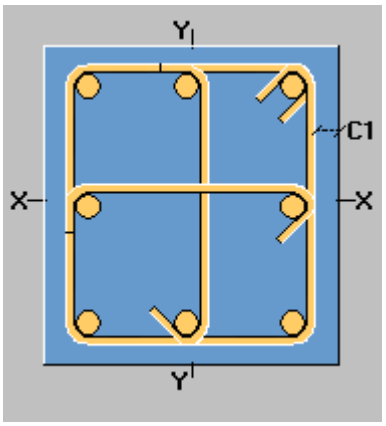
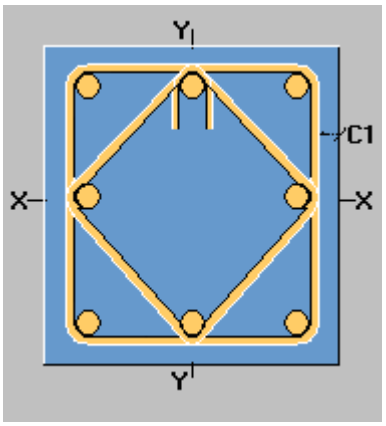
Reinforcement pattern

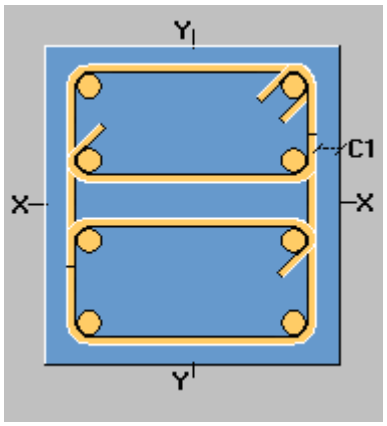
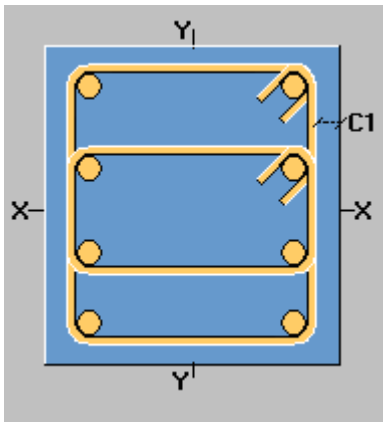
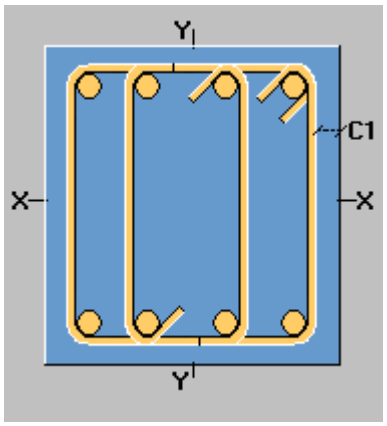
Primary ties are distributed along the entire length of the column.

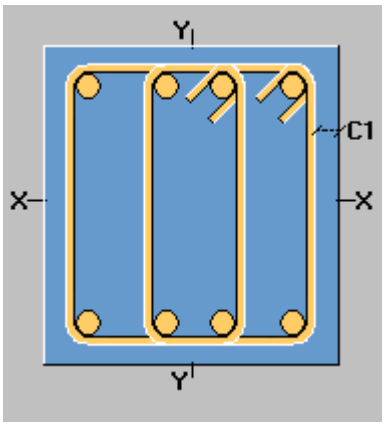
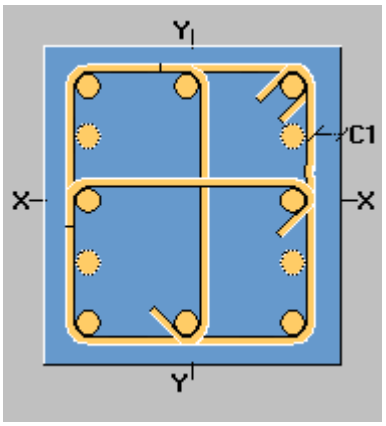
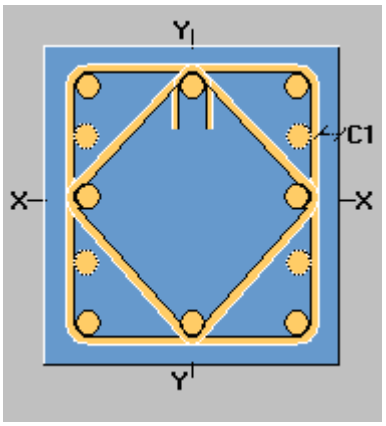
First select the desired tie reinforcement pattern from the list box. There are 16 predefined patterns to choose from. The longitudinal reinforcing bar selection will affect which of these patterns can be used. The component will verify that the primary tie configuration correlates with the primary longitudinal reinforcement configuration. Furthermore, the location or stirrup in section may depend on the location of longitudinal bars (e.g. 8-bar X-X).

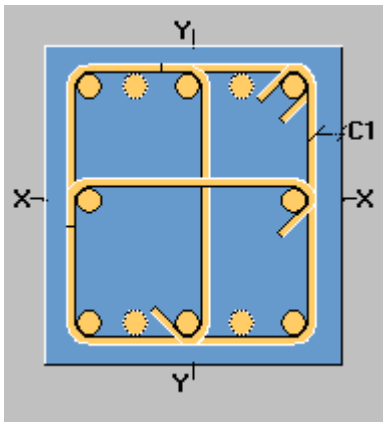
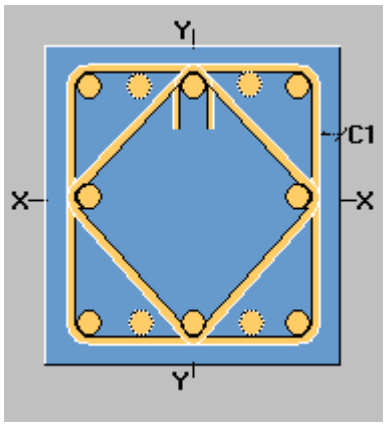
Description	Option
4 Bar Symmetrical	
6/10 Bar X-X (A)	

Description	Option
6/10 Bar X-X (B)	
6/10 Bar Y-Y (A)	
6/10 Bar Y-Y (B)	

Description	Option
8 Bar Symmetrical (A)	
8 Bar Symmetrical (B)	
8 Bar Symmetrical (C)	

Description	Option
8 Bar X-X (A)	
8 Bar X-X (B)	
8 Bar Y-Y (A)	

Description	Option
8 Bar Y-Y (B)	
12 Bar X-X (A)	
12 Bar X-X (B)	

Description	Option
12 Bar Y-Y (A)	
12 Bar Y-Y (B)	

Properties

Option	Description
Tie size	Select the size of the tie bar.
Tie grade	Select the grade of the tie bar.
Hook type	Select the tie hook type. The options are 90-degree and 135-degree hooks.
c1	Specify the minimum clear cover. The component must verify that with the clear cover specified the tie properly fits around the longitudinal reinforcing bar with the locations specified in previous tab.
d1	Specify the distance to the centerline of the first tie from the top and bottom. Often there are anchor bolt

Option	Description
	pockets at the bottom of columns. The component should interpret the **d1** distance to be above those pockets.
X	Specify the quantity.
Spacing	Define the reinforcing bar spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.
Bars	Define the number of reinforcing bars.
Same as left end	Define whether the same spacing and reinforcing bar properties are used at the right end.

Secondary reinforcement tab

Use the **Secondary reinforcement** tab to define the longitudinal bar and ties used in the secondary section(s) of the column.

The component will automatically determine these sections from the column geometry.

Option	Description
Longitudinal bar size	Select the bar size for the secondary longitudinal bars.
Longitudinal bar grade	Enter the grade for the secondary longitudinal bars.
Tie size	Select the bar size for the secondary ties.
Tie grade	Enter the grade for the secondary ties.
Min. clear cover **c2**	Specify the minimum clear cover from the secondary longitudinal bars to the column face.
End clear distance	Specify the end clear distance from the top and bottom of each secondary section to the ends of the secondary longitudinal bars.
Hook type	Select the secondary tie hook type. The options are 90-degree and 135-degree hooks.

Option	Description
Spacing	Define the reinforcing bar spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.
Bars	Define the number of reinforcing bars.
Same as left end	Define whether the same spacing and reinforcing bar properties are used at the right end.

Circular column tab

Use the **Circular column** tab to define the reinforcement parametrics for circular columns.

The component will only create reinforcement in constant diameter members. Tapered columns and circular columns with ledges are not supported.

Option	Description
Number of main bars	Specify the desired number of main bars in the column, from 4-12.
Main bar size	Select the size of the main bar.
Main bar grade	Select the grade of the main bar.
Tie size	Select the size of the tie bar.
Tie grade	Select the grade of the tie bar.
c1	Specify the tie clear cover.
End clear distance	Specify the end clear distance from the top and bottom of each secondary section to the ends of the secondary bars.
d1	Specify the top end distance from the centerline of the first tie to the top of the column.
d2	Specify the bottom end distance from the centerline of the last tie to the bottom of the column. Often there are anchor bolt pockets at the bottom of columns. The component should interpret the **d2** distance to be above those pockets.

Option	Description
X, **s1**	Specify the quantity, **X**, and spacing, **s1**, of the first group of ties at top and bottom.
Max. spacing	Specify the maximum spacing, **s2**, of primary ties throughout the length of the column.
Bars	Define the number of reinforcing bars.
Spacing	Define the reinforcing bar spacing. Use a space to separate spacing values. Enter a value for each space between the reinforcing bars. For example, if there are 3 reinforcing bars, enter 2 values.

Attributes tab

Use the **Attributes** tab to define naming and numbering of the reinforcement.

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Minimum cut size	Define the minimum length of the boundary of cut/hole, to be considered as cut in this field. The default value is 200 mm.

Strand code

Store strand code – With option **Store strand code** you may choose to store (or not) a user-defined textual attribute `TS_STRAND_CODE` to part.

Strand code – In field **Strand code** you may enter a string template for the value of the above mentioned user-defined attribute. The default template is the total number of created strands. The template may contain any text and the predefined fields for:

- %NUMBERS – is the numbers of created strands with different sizes and/or grades e.g. 10 (for one size & grade) or 6+4 (if two sizes and/or grades has been used).
- %SIZES – sizes of the strands e.g 1/2 or 1/2 + 3/8.
- %GRADES – grades of strands.

Strand creation at voids

Create strand at voids – The option **Yes** will show the strands even though the voids are present. The option **No** will not show the strand if void is present at any location along the strand path.

Select to cut the strands at void locations from **Cut strands at voids**.

Topping (58)

Use the **Topping (58)** component to model typical toppings along with their typical reinforcement.

Topping tab

Use the **Topping** tab to define the method of topping, configuration of elevation/thickness of topping, and the mesh created in the topping.

Option	Description
Elevation is specified	Select the method to use to specifiy the elevation of the exterior boundary surface of the topping: • **Absolute elevation**, EL: you can to specify the top surface elevation relative to elevation 0'-0". • **Relative thickness**, H: you can specify the top surface as a uniform thickness above the underlying precast floor units. • **Use control points**: Tekla Structures uses the polygon input points as the actual 3 dimensional nodes of the top surface of the toppings. Use this option to place a topping over any sloped surface.
Elevation value, EL	Enter an appropriate value based on the choice picked in the list box

Option	Description
	above. This input box is grayed out if you have selected the **Use control points** option.
Topping configuration	Specify whether the topping is integral with the underlying precast unit (monolithic) or poured on site.
Concrete material	Specify the concrete strength for the topping, from the material catalog.
Reinforcing mesh	Select the mesh reinforcement from the mesh catalog.

Advanced tab
Use the **Advanced** tab to alter the elevation of individual nodes.

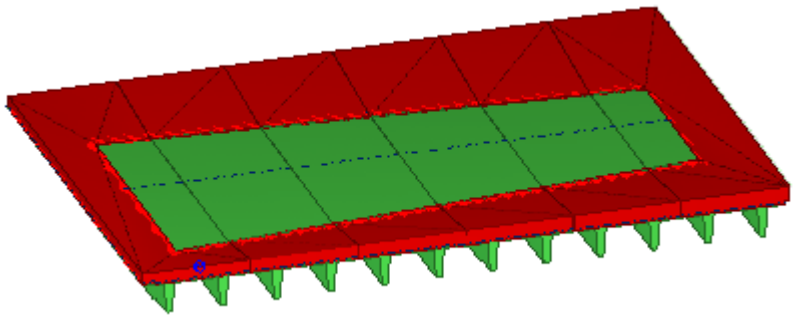
Option	Description
Control point indexes	Specify one control point or multiple control point indexes simultaneously.
Thickness	Specify the actual elevation/thickness (thickness at individual control points).

Attributes tab
Use the **Attributes** tab to define mesh and concrete properties.

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Wash (59)
Use the **Wash (59)** component to model typical washes along with their typical reinforcement.

Use for

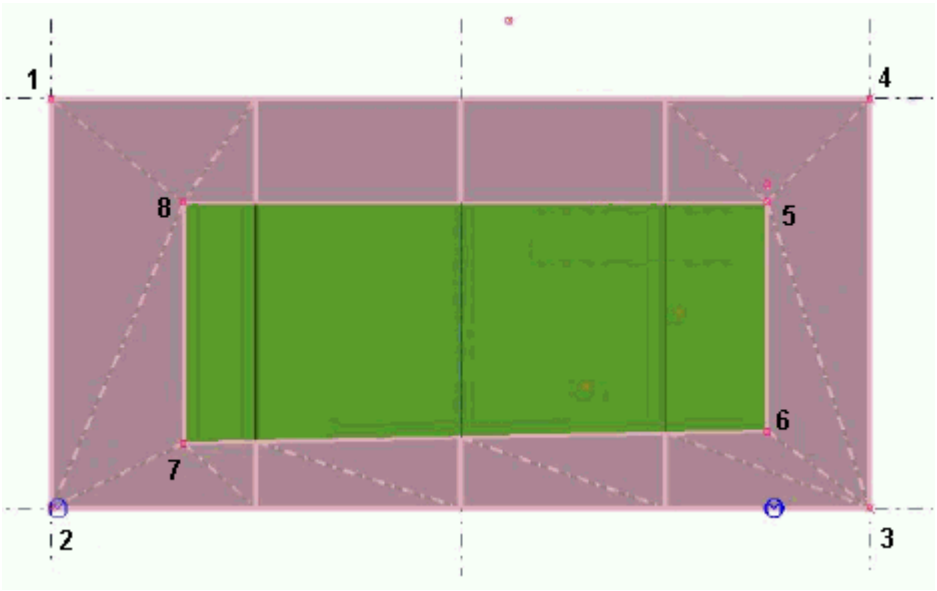


Before you start

Create slabs using the **Create concrete beam** command or with a slab component.

NOTE You cannot use the **Wash (59)** component in slabs that are created as a polygon plates, such as slabs created with **Create concrete slab** command.

Picking order



1. Select the slab objects. If the slabs are created with a component, activate the **Select objects in components** switch before selecting slabs.
2. Click middle mouse button.
3. Pick points counterclockwise to indicate external boundary (points 1 to 4 in the picture above).
4. Click middle mouse button outside the external boundary.

5. Pick points clockwise to indicate internal boundary (points 5 to 8 in the picture above).
6. Click middle mouse button somewhere along the internal boundary.

Wash tab

Use the **Wash** tab to define the method of wash, configuration of the elavation/thickness and size of wash, recess depth, and mesh to be created in the wash.

Specify the wash step dimensions, **D1** and **D2**, to define the top elevation of the internal boundary relative to the surface of the precast floor units. Note that D1 and/or D2 can be zero (0.00).

Option	Description
Elevation is specified	Select the method to use to specifiy the elevation of the exterior boundary surface of the wash: • **Absolute elevation**, EL: you can to specify the top surface elevation relative to elevation 0'-0". • **Relative thickness**, H: you can specify the top surface as a uniform thickness above the underlying precast floor units. • **Use control points**: Tekla Structures uses the polygon input points as the actual 3 dimensional nodes of the top surface of the wash. Use this option to place a wash over any sloped surface.
Elevation value, EL	Enter an appropriate value based on the choice picked in the list box above. This input box is grayed out if you have selected the **Use control points** option.
Topping configuration	Specify whether the wash is integral with the underlying precast unit (monolithic) or poured on site.
Concrete material	Specify the concrete strength for the wash from the material catalog.

Option	Description
Reinforcing mesh	Select the mesh reinforcement from the mesh catalog.

Advanced tab
Use the **Advanced** tab to define the alter the elevation of individual nodes.

Option	Description
Exterior control point indexes	Specify one control point or multiple control point indexes simultaneously.
Thickness	Specify the actual elevation/thickness (thickness at individual control points).
Interior control point indexes	Specify one control point or multiple control point indexes simultaneously.
D1	Change the thickness at each interior control point.
D2	Change the recess depth at individual control point.

Attributes tab
Use the **Attributes** tab to define the required mesh and concrete properties.

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Hollowcore reinforcement strands (60)
The Hollowcore “Automated Reinforcement Generator” (HC-ARG) component enables Tekla Structures to accurately model all typical hollowcore reinforcement. Use the tabs in the component dialog box to load a previously saved HC (Hollowcore) reinforcement scheme, or new reinforcement scheme.

Strand template tab

Use the **Strand template** tab to define the number and arrangement of vertical strand planes in each stem, and the vertical distances between grid points in each plane.

Define distances

You can select to use either the values defined in the component dialog box (**Use dialog data**), or you can select to use an external `.dat` file (**Use external file**).

- **Use dialog data**: The component uses the grid points defined in **Grid spacing along X**, and **Grid spacing along Y**.
- **Use external file**: You can add additional grids for strand patterns in an external `.dat` file. You can find an example `.dat` file called `HollowcoreReinforcementStrands.dat` in the common environment.

Grid spacing

Option	Description
Grid spacing along X	• Use grid spacing to define grid spacing values for the X axis. Separate the values with commas or spaces. • You can use multiplier values to define a repeating series of values, for example, 8*2 indicates 8 holes with a 2 unit spacing. • The first value is the distance from the left edge of the plank. Each successive value is the distance to the next grid point.
Grid spacing along Y	• Use grid spacing to define grid spacing values for the Y axis. Separate the values with commas or spaces. • The first value is the distance from the bottom edge of the beam. Each successive value is the distance to the next grid point.

Strand pattern tab

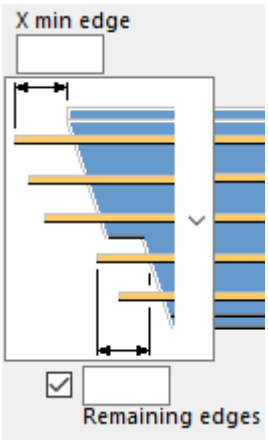
Use the **Strand pattern** tab to define the actual strand properties.

Option	Description
Row	Define the row number in the strand template for the start of the strand.
Columns	Define a list of one or more column numbers and/or column ranges in the strand template at which a strand should be created.
Size	Define the size of the strands by selecting it from the reinforcing bar catalog.
Grade	Define the strand grade.
Pull	Define the pull force of the strands. You can enter a pull value even when the distances are defined by an external `.dat` file. If you do not enter a pull value, the values in the `.dat` file are used.

Strand extension tab

Use the **Strand extension** tab to define the strand extensions.

You can define the extensions separately for the longest edge and the remaining edges.



Attributes tab

Use the **Attributes** tab to define the naming and numbering properties of the strands.

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.

Option	Description
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Strand code

Store strand code – With option **Store strand code** you may choose to store (or not) a user-defined textual attribute `TS_STRAND_CODE` to part.

Strand code – In field **Strand code** you may enter a string template for the value of the above mentioned user-defined attribute. The default template is the total number of created strands. The template may contain any text and the predefined fields for:

- %NUMBERS – is the numbers of created strands with different sizes and/or grades e.g. 10 (for one size & grade) or 6+4 (if two sizes and/or grades has been used).
- %SIZES – sizes of the strands e.g 1/2 or 1/2 + 3/8.
- %GRADES – grades of strands.

Strand creation at voids

Create strand at voids – The option **Yes** will show the strands even though the voids are present. The option **No** will not show the strand if void is present at any location along the strand path.

Select to cut the strands at void locations from **Cut strands at voids**.

Post-tensioning (61)

Post-tensioning (61) component enables Tekla Structures to accurately model most post-tensioning profiles and attributes. Use the tabs in the component dialog box to load a previously saved PT (post-tensioning) tendon scheme, or a new tendon scheme.

Tendon properties tab

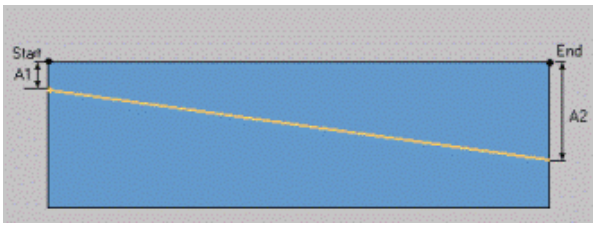
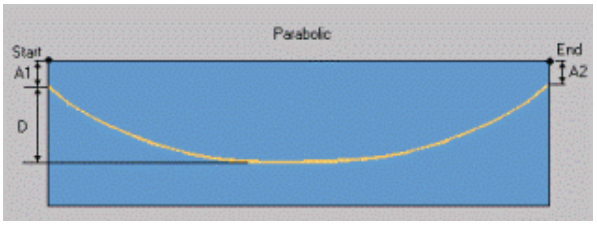
Use the **Tendon properties** tab to define tendon properties, the number and spacing of tendons, rotation angle, and grouted system.

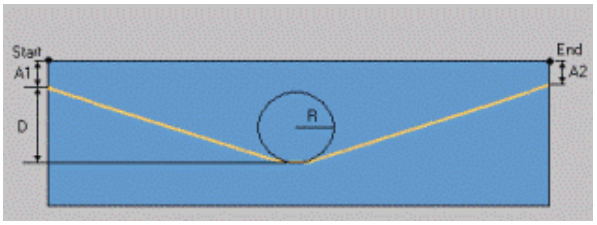
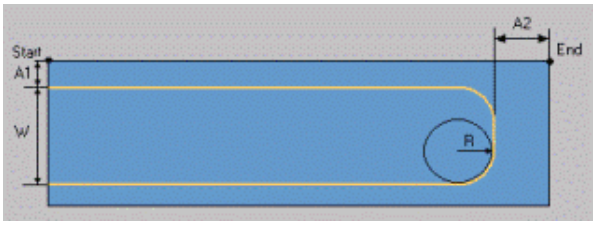
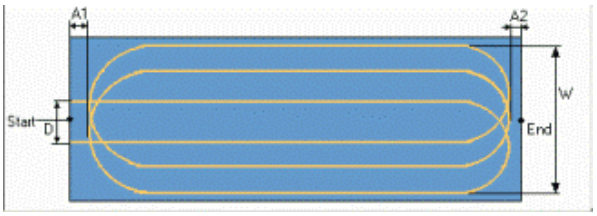
Option	Description
Tendon size and grade	Select the tendon size and grade from the catalog of available options.

Option	Description
Number of tendons	Enter the number of tendons in the group.
Spacing between tendons, S	If you have specified more than one tendon, input the spacing (S) between tendons.
Rotation angle	Specify the rotation angle of tendon/tendon group around the input points from the default plane. The default plane is the plane which is parallel to the“strong axis”of the part.
Grouted system	Specify whether the tendon is to be a grouted system or a non-grouted system. This option defines whether Tekla Structures uses the strand in determining the ultimate capacity. Furthermore, you can input custom or vendor specific post-tensioning information, including unique and/or multiple tendon systems, grease thickness, jacket material, jacket thickness, and jacket diameter.

Tendon profile tab

Use the **Tendon profile** tab to define the tendon profile, offsets, bend radius, width, and loop count.

Option	Description
	Straight
	Parabolic

Option	Description
	Straight one point
	Loop
	

Depending on the profile selected, you can input the required fields below:

- **Start point offset A1**
- **End point offset A2**
- **Sway drape D**
- **Bend radius R**
- **Width W**
- **Centerline Offset**
- **Loop count**

Attributes tab

Use the **Attributes** tab to define the naming and numbering properties of the tendons.

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.

Option	Description
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

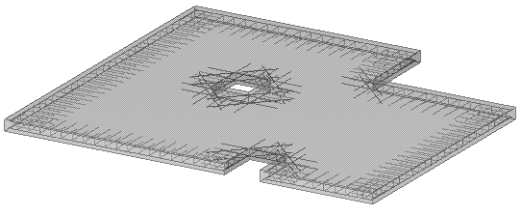
Edge and Corner Reinforcement (62)

Edge and corner reinforcement (62) creates edge and corner reinforcement for concrete panels and slabs. You can reinforce solid or sandwich precast concrete panels, or vertical and horizontal cast-in-situ slabs.

Objects created

- Reinforcement

Use for

Situation	Description
	Slab with edge and corner reinforcement and diagonal bars around openings

Limitations

Edge and corner reinforcement (62) does not create the slab main reinforcement. The main reinforcement can be created using slab reinforcement components, for example, **Pilecap reinforcement (76)**, **Pad footing (77)**, **Slab bars (18)**, or **Reinforcement mesh array in area (89)**. The main reinforcement can also be created by using the reinforcement creation commands on the **Rebar** tab on the ribbon.

Before you start

Create a panel or a slab.

Selection order

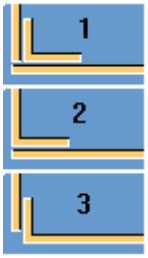
1. Select the main part (panel or slab).

   The reinforcement is created automatically.

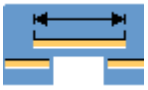
Picture tab

Use the **Picture** tab to control the edge and corner reinforcements, the cutouts, and the corner cutouts.

Reinforcement

Option	Description
	Define how the corners are reinforced.
	Diagonal corner bars are created to an opening.
	Diagonal corner bars are not created to an opening.
Aligned with slab slope	Select whether or not the edge bars, diagonal bars and U bars are aligned with the slope of a slab that has different corner thicknesses. The default value is **Yes**.

Cutouts

Option	Description
	Edge bars are not created to the cutout.
	Edge bars are created above the cutout for the defined length.
	Edge bars are created above the cutout.
	Edge bars are created to the cutout.
Reinforce opening	Select whether openings are reinforced.
Max block cut size	Define the size limit for the cutouts that affects the edge and corner reinforcement and the diagonal bar creation. Edge and corner reinforcement are not created to cutouts that are smaller than the defined size. Diagonal bars are not created to cutouts that are smaller than the defined size.

Corner cutouts

Option	Description
	Edge bars on both sides of the corner cutout are bent. This option overrides the **Corner bar** options 1, 2, and 3.
	Both the edge bars and the U bars are moved a distance equal to the length of the shorter side of the cutout.
	No bars are moved. Edge bars are created as if there was no cutout.
Max corner cut out size	Define the size limit for the corner cutouts. Edge or corner reinforcement are not created to cutouts that are smaller than the defined size.
Min hole size	Define a size limit for the holes. Edge and corner reinforcements are not created around holes that are smaller than the size limit. If the hole is not rectangular or round, the size of the hole is determined by the maximum dimension of the bounding box around the hole. For example, if you do not want to create edge or corner reinforcement around holes that are 200 mm or smaller, enter 200.

Edge/Diag bars tab

Use the **Edge bars** and **Diag bars** tabs to control the edge and diagonal reinforcing bar properties, reinforcing bar overlapping, and concrete cover thickness.

Reinforcing bar properties

Option	Description
Prefix	Prefix for the part position number.
Start number	Starting number for the part position number.
Name	Name that is shown in drawings and reports.
Size	Diameter of the reinforcing bar.
Grade	Strength of the steel used in reinforcing bars.
Bending radius	Edge bar bending radius.

Option	Description
Class	Use the **Class** to group reinforcing bars. For example, you can display reinforcing bars of different classes in different colors.

Reinforcing bar overlap

Option	Description
L1, L2, L3	Reinforcing bar overlapping length.
Number of reinforcing bars	Number of reinforcing bars in the reinforcing bar group.
Create as one group	Select whether the identical edge bars and diagonal bars are created as a group. The default value is **Yes**.

Concrete cover thickness

Option	Description
C1, C2, C3, C4	Bottom, top, and side concrete cover thickness for the edge and the diagonal bars.

Minimum bar length

Option	Description
Minimum bar length	Minimum length of the edge bars on the **Edge bars** tab. If the bar is shorten than the entered value, the bar is not created.

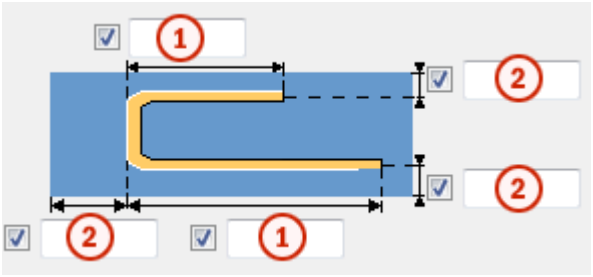
Additional edge bars

Option	Description
Additional edge bars	Select **Yes** to create additional secondary edge bars on the **Edge bars** tab. Define the offset for the bars.

Horizontal/Vertical U bars tab

Use the **Horizontal U bars** and **Vertical U bars** tabs to control the U bar leg length, concrete cover thickness, and reinforcing bar properties and spacing.

U bar length



Option	Description
1	U bar leg length.
2	Top, bottom, and side concrete cover thickness for U bars.

Reinforcing bars

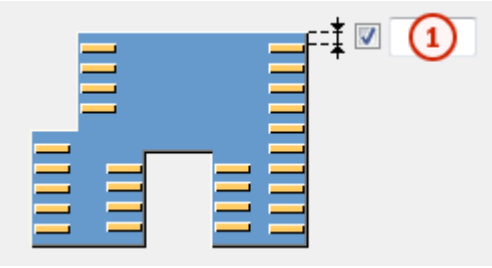
Option	Description
Create U bar	Select whether the U bars are created. In addition, on the **Vertical U bars** tab, you can select whether the vertical U bars are created using the same settings as the horizontal U bars. **Same as horizontal and inside**: vertical U bars use the same reinforcing bar settings and concrete cover value as the horizontal U bars, and the vertical U bars are placed so that they fit inside the horizontal U bars. **Same as horizontal and outside**: vertical U bars use the same reinforcing bar settings and concrete cover value as the horizontal U bars, and the vertical U bars are placed outside the horizontal U bars.
Prefix	Prefix for the part position number.
Start number	Starting number for the part position number.
Name	Name that is shown in drawings and reports.
Size	Diameter of the reinforcing bar.
Grade	Strength of the steel used in reinforcing bars.
Class	Use the **Class** to group reinforcing bars. For example, you can display reinforcing bars of different classes in different colors.
Bending radius	U bar bending radius.

U bar creation and spacing

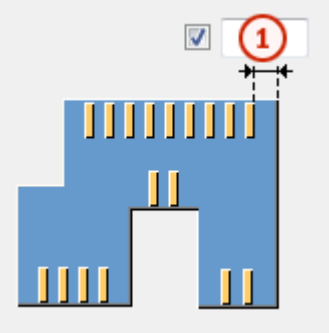
Option	Description
Ignore cuts smaller than	Cuts that are smaller than the entered value are ignored and do not affect the U bar placement.
Creation method	Select how the U bars are distributed.
Number of reinforcing bars	Number of U bars in the reinforcement. This field is available if you set the **Creation method** to **Equal distribution by number of reinforcing bars**.
Target spacing value	Target spacing value. Tekla Structures tries to distribute the U bars as closely as possible according to the entered value. This field is available if you set he **Creation method** to **Equal distribution by target spacing value**.
Exact spacing value	Exact spacing value. Tekla Structures creates fixed, equal spaces between the reinforcing bars.
By exact spacing	Define several spacing values between the U bars. Tekla Structures distributes the bars using the values you enter in the field. Enter all spacing values, separated by spaces. You can use multiplication to repeat spacing values. For example, enter 5*200 to create five spaces of 200 mm.

End distance

Horizontal U bars:

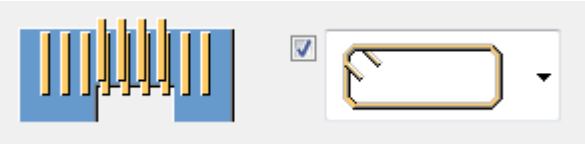


Vertical U bars:



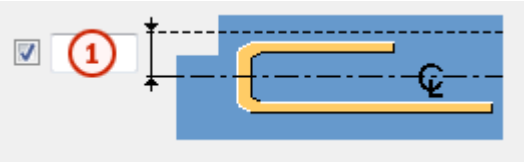
Option	Description
1	U bar distance from the panel or the slab end.

Stirrups



Stirrups are created instead of U bars when the distance between the edges in the panel or the slab is so small that the U bars do not get the required cover thickness.

Position of scanning plane



Option	Description
1	U bar offset.

U Bar of concrete slab (63)

U Bar of concrete slab (63) automatically generates U bars reinforcement for concrete slabs.

Reinforcements

- Precast concrete RC slabs and PC panels (both solid and sandwich)
- For cast-in-situ slabs (both vertical and horizontal).

Use for

To create U bars to an already created concrete slab complete with its main and secondary reinforcement.

Create the slab's main reinforcement

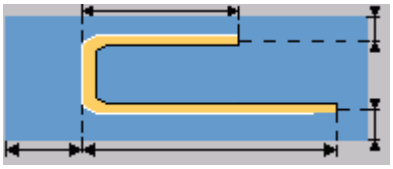
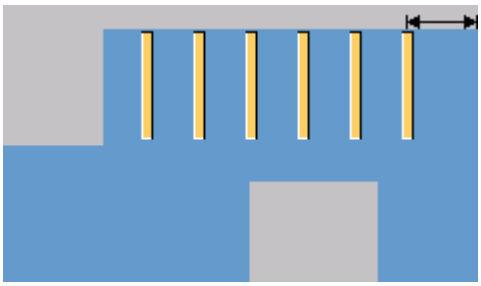
- with existing slab reinforcement components (e.g. **Pilecap reinforcement (76)**, **Pad footing (77)**, **Slab bars (18)**, **Reinforcement mesh array in area (89)**)
- manually with the tools on the **Concrete** toolbar (e.g. single bar, bar groups or mesh).

Before you start

Existing components must be created previously in order to use this reinforcement tool.

U bars tab

Use the **U bars** tab to define the U bar properties.

Option	Description
	U Bar UB1
	U Bar UB2

Use the fields to specify

- Top concrete cover.
- Bottom concrete cover.
- Side concrete cover.
- Top leg length.
- Bottom leg length.
- End dist

Option	Description
Prefix	Prefix for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Size	Diameter of the reinforcing bar.

Option	Description
Grade	Strength of the steel used in the reinforcing bars.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Bending radius	Internal radius of the bends in the bar.
Creation method	Spacing options for the bars.
Target spacing value	Target spacing value. Tekla Structures tries to distribute the U bars as closely as possible according to the entered value. This field is available if you set he **Creation method** to **Equal distribution by target spacing value**.
Exact spacing value	Exact spacing value. Tekla Structures creates fixed, equal spaces between the reinforcing bars.
By exact spacing	Define several spacing values between the U bars. Tekla Structures distributes the bars using the values you enter in the field. Enter all spacing values, separated by spaces. You can use multiplication to repeat spacing values. For example, enter 5*200 to create five spaces of 200 mm.

Wall panel (64)

Wall panel (64) creates reinforcement for different wall panels. Contains options to create strands and stirrups. Is suitable also for insulated wall panels. You can easily change the direction (horizontal or vertical) of most reinforcing bars. Automatically creates parallel or diagonal corner reinforcement around the openings and notches.

Objects created

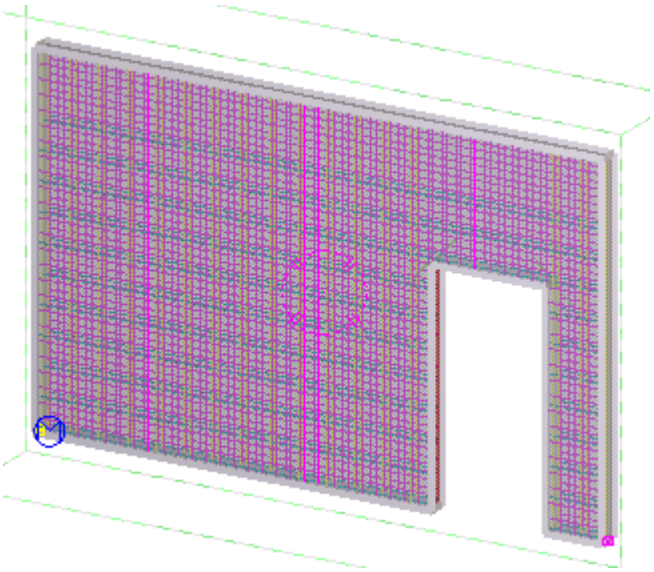
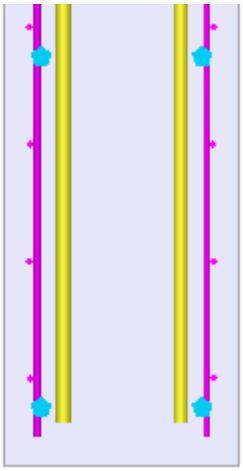
This component contains the following reinforcement groups:

- Strands

- Longitudinal reinforcing bars
- Transverse mesh
- Transverse straight reinforcing bars
- Transverse closed stirrups
- Reinforcement for openings

To omit any of these reinforcement groups, set the reinforcing bar size or the mesh grade to zero.

Use for

Situation	Description
	Reinforcement created with default settings for a wall panel with a door opening.
	A cross section from the wall reinforcement.

Limitations

Does not create reinforcement for the ledges in the wall.

Before you start

Create the wall panel.

Selection order

1. Select the wall panel.

Strand template tab

Use the **Strand template** tab to define the strand positions with grid points within the wall cross section. Strands may only be located at the positions defined in this template, but the template allows for a mixture of regular and irregular points.

Strand at opening creation

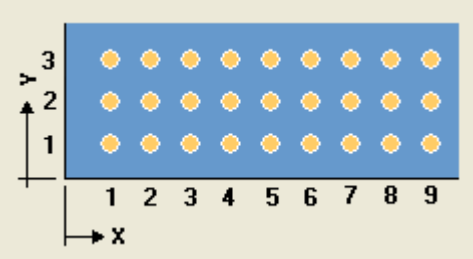
Select to create strands at the opening or not.

Strand direction

Changes the direction of strands in the wall.

Grid Spacing along X

Use to define grid spacing values for the x axis. Separate values with commas or spaces. The first value is the distance from the left edge of the spandrel. Each successive value is the distance to the next grid point.



TIP Use multiplier to define a repeating series of values. For example, 8*2 indicates eight holes with a two unit spacing.

Grid Spacing along Y

Use to define grid spacing values for the y axis. Separate values with commas or spaces. The first value is the distance from the bottom edge of the spandrel. Each successive value is the distance to the next grid point.

Strand pattern tab

Use the **Strand pattern** tab to create strands at the grid positions defined by a strand template.

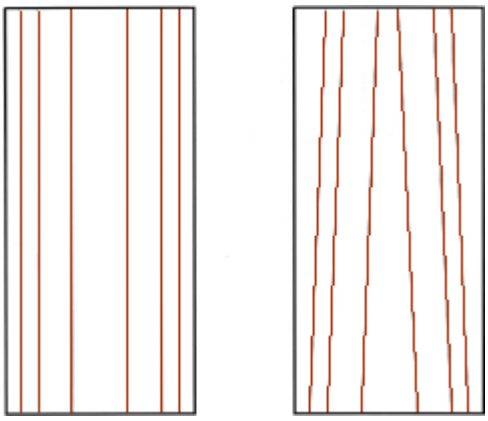
Same as left

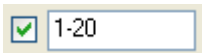
Use to simplify strand input.

- Select **Yes** to use the row and debond information of the left end for the strands at the right end of the spandrel.
- Select **No** to specify alternate row and debond information at each end of a strand.

Strand pattern definitions

Use the strand template positions in the strand grid to specify the row and column for each end of the strand. Strands may have different grid row positions at each end, but a strand may not cross grid columns.



Field	Description
Left end - Column	Enter the vertical row number in the strand template for the left end of the strand.
Left end - Debond	Enter the debond length at the left end of the strand.
Right end - Column	If **Same as left** is **No**, enter the vertical row number in the strand template for the right end of the strand.
Right end - Debond	If **Same as left** is **No**, enter the debond length at the right end of the strand.
Rows	Enter a list of one or more row numbers and/or row ranges in the strand template at which a strand should be generated. 
Size	Select the size of the strands.
Grade	Select the strand grade.
Pull	Enter the pull force of the strands.

Longitudinal rebar tab

Use the **Longitudinal rebar** tab to specify longitudinal mild reinforcement for the wall panel.

Minimum end clear distance C & D

Specify minimum end clearance distance for C and D fields.

Rebar direction

Use to change the direction of the longitudinal reinforcing bars.

Length type

Select a **Length type** to define the end constraints of the reinforcing bar group within the member length. Depending on your choice, enter additional information is in the **L**, **C**, or **D** fields.

Option	Description
Full length	Reinforcing bar extends the full length of the wall panel – excluding the minimum end clear distance at each end. The **L**, **C**, and **D** fields are disabled.
Centered	Specify a fixed length **L** for the bar which the component centers in the length of the wall panel. The **C** and **D** fields are disabled.
Known L, C	Specify a fixed length **L** for the bar and a distance **C** from the start of the wall panel. The **D** field is disabled.
Known L, D	Specify a fixed length, **L** for the bar and a distance **D** from the end of the wall panel. The **C** field is disabled.
Known C, D	Specify a distance **C** and **D** from the end of the wall panel. The **L** field is disabled.

XLoc

Specify the horizontal locations of the longitudinal reinforcing bars.

YLoc

Specify the vertical location of the longitudinal reinforcing bars.

Mesh tab

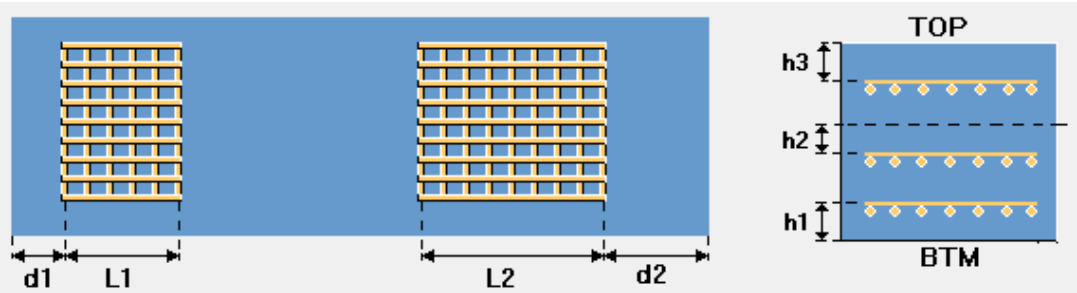
Use the **Mesh** tab to specify mesh for reinforcement in the wall panel.

Mesh properties

Select the mesh from the catalog and Tekla Structures fills the other fields automatically.

Layer properties

You can specify one to three different layers of mesh. Each mesh can be different or identical.

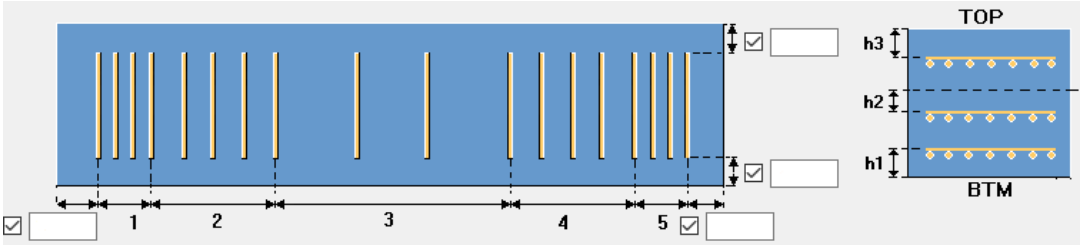


Option	Description
Side cover	Concrete cover thickness.
Ht to layer	Specify height to the center line of each layer – h1, h2, h3. If you leave this field blank, no mesh is created.
Left end distance	Specify clearance distance for the left end. This is the dimension **d1** in the pictureimage above.
Right end distance	Specify clearance distance for the right end. This is the dimension **d2** in the image above.
Continuous	Specify whether the mesh is continuous over the wall panel length, or is used only at the end(s) of the wall. The options are: • Select **Yes**: Mesh extends the full length of the member • Select **No**: Mesh can be defined at each end using the **L1** and **L2** fields. If any openings are present, the component will trim the mesh around the opening using the end distance (**d1**) dimension as a clearance.
Left end length	If the mesh is used only at the ends of the wall, enter the length for the left end. (**L1**)
Right end length	If the mesh is used only at the ends of the wall, enter the length for the right end. (**L2**)
Cross bar location	Defines whether the crossing bars are located above or below the longitudinal bars.

Transverse Straight bars tab

Use the **Transverse straight bars** tab to specify transverse straight reinforcing bars with varying spacing in the wall panel.

Properties of wall transverse straight bars



- Specify the side clear cover.
- Specify the end distance to the center line of the first reinforcing bar from each end.
- Specify the quantity and spacing of reinforcing bars in zones next to each end.
- If you need another group of reinforcing bars at each end, specify the quantity and spacing of reinforcing bars in the second group. If the second group is not required, enter `0` in the quantity field.
- If reinforcing bars are required to extend the length of the wall panel, specify the maximum spacing allowed between end groups – **Spacing** in the third group. If **Spacing** is left blank, there is no additional reinforcing bar between the end groups.

NOTE At openings, this component adds reinforcing bars at each side of the opening using the side cover dimension as distance from the opening to the center line of the reinforcing bar. These reinforcing bars are in addition to the typical reinforcing bar.

Option	Description
Spacing	Define the reinforcement bar spacing.
Bars	Define the number of reinforcing bars.
Rebar size	Diameter of the reinforcing bar.
Same as left end	Select whether the same properties are used at the right end.
Rebar grade	Strength of the steel used in the reinforcing bars.
h1, **h2**, **h3**	Specify height to reinforcement level. If **h1**, **h2**, or **h3** is left blank, no reinforcing bar is created at that level.

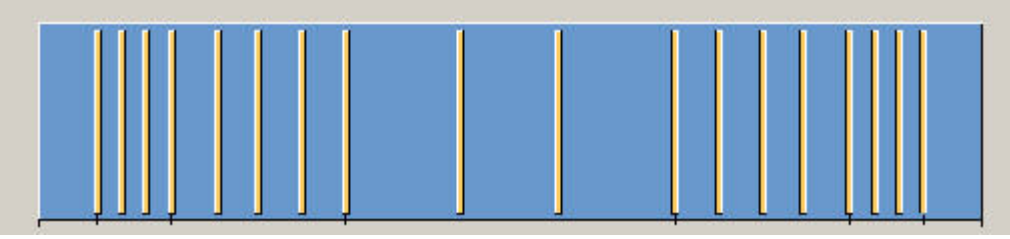
Option	Description
Bars direction	Change the direction of the bars.

Transverse stirrups tab

Use the **Transverse stirrups** tab to define the distribution of the stirrups along the wall panel. You can define up to five zones for the distribution. If the selected wall panel is an insulated wall, Tekla Structures details the stirrups in the structural (thicker) wythe only. If both wythes are the same thickness, Tekla Structures details the stirrups in the same way in both wythes.

Stirrup properties

For each zone you can specify a different spacing of stirrups. Specify the end distance to the center line of the first stirrup from each end.

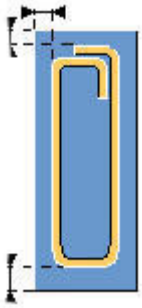
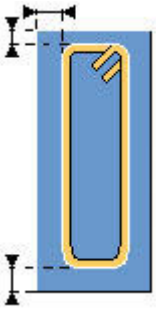
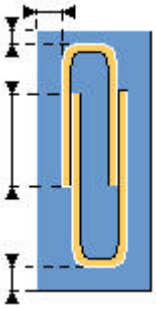


- Define the spacing between two stirrups in each zone.
- Define the number of reinforcing bars in each zone.
- Select the bar size.

Stirrup type

Select the stirrup type and enter needed dimensions.

Option	Description
	No stirrups

Option	Description
	90-degree closed stirrup • Specify the side clear cover. • Specify the top and bottom clear cover.
	135-degree closed stirrup • Specify the side clear cover. • Specify the top and bottom clear cover.
	U bar pairs • Specify the side clear cover. • Specify the top and bottom clear cover. • Specify the lap length

NOTE
- The component creates stirrups using the minimum CRSI bend diameter.
- At openings, this component adds a stirrup/bar each side of the opening using the top cover dimension as distance from the opening to the center line of the stirrup/bar. These stirrups are in addition to the typical stirrups/bar.

Same as left end

Set this to **Yes** if stirrups are symmetric at both ends of the wall panel.

Rebar grade

Define the grade of the transverse stirrups.

Stirrups direction

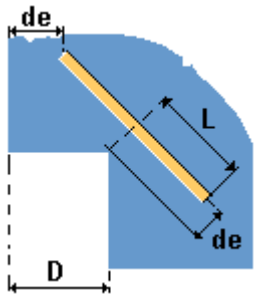
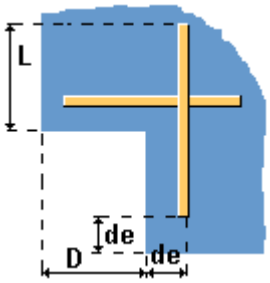
Select an option to define the stirrup direction.

Notch/Opening tab

Use the **Notch/Opening** tab to specify reinforcement around different openings in a wall panel.

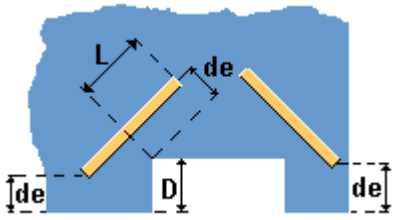
Corner notch reinforcement

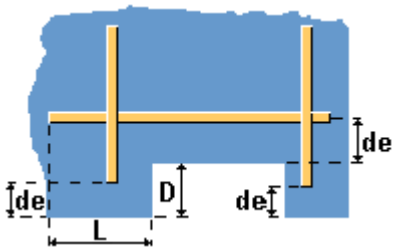
Select a reinforcement scheme to apply around wall panel corner notches. The options are:

Option	Description
	Diagonal reinforcement around the corner notches.
	Parallel reinforcement around the corner notches.

Side/end notch reinforcement

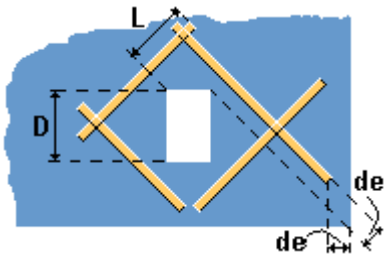
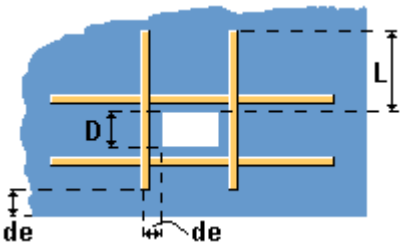
Select a reinforcement scheme to apply around the side and end notches of the wall panel. The options are:

Option	Description
	Diagonal reinforcement around the side or end notch.

Option	Description
	Parallel reinforcement around the side or end notch.

Opening reinforcement

Select a reinforcement scheme applied around openings. The options are:

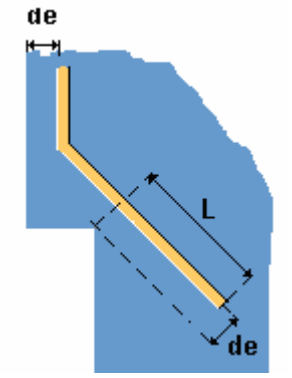
Option	Description
	Diagonal reinforcement around the opening.
	Parallel reinforcement around the opening.

Notch/opening reinforcement properties

Field	Description	Default
Min notch dimension, D	Specify first the minimum dimension D beyond which reinforcement is applied. For example, if a D value of 2.00 is specified, component will not apply reinforcement parallel to that direction.	6" (150 mm)
Edge clearance, de	Specify the edge clearance from the notch and edge of the wall panel.	1,50" (38 mm)

Field	Description	Default
Extension, L	Specify the extension of the reinforcing bar beyond a notch or re-entrant corner. This component checks that the L dimension specified is at least the minimum development length of the bar selected. If the L dimension specified cannot be achieved due to edge clearances, hooks are added automatically.	18" (450 mm)
Rebar size	Defines the size of the reinforcing bars around notches or openings. Click ... button to select a reinforcing bar from the reinforcing bar catalog. Selecting a size will override the value in the **Rebar grade** field.	#4 (M13)
Rebar grade	Field for defining the grade of the reinforcing bars. This field cooperates with the **Rebar size** field.	A615

If any of the straight or diagonal bars approaches closer to the concrete edge than **de** from the edge given L, the bar is bent as follows:



Attributes tab

Use the **Attributes** tab to define the naming and numbering properties.

Option	Description
Prefix	Prefix for the part position number.

Option	Description
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

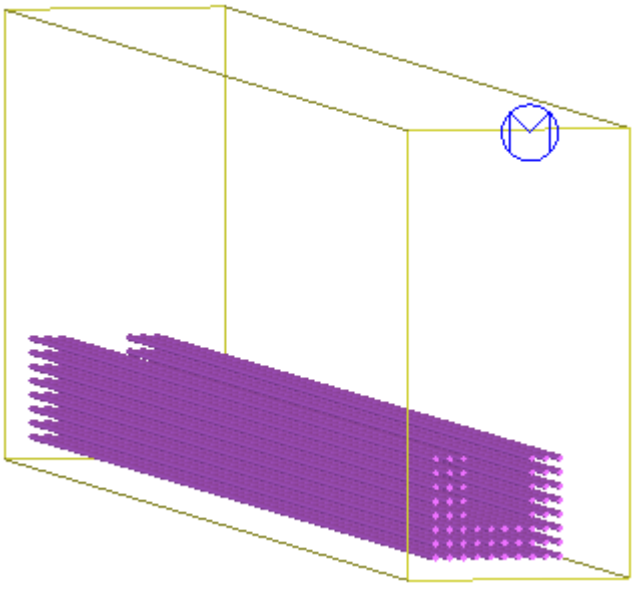
Reinforcement Strand Layout (66)

Reinforcement Strand Layout (66) creates straight or depressed prestress strands for flexural reinforcement in a beam.

Objects created

- Straight and depressed prestress strands.

Use for

Situation	Description
	Straight strands.

Selection order

1. Select the beam.

Strand template tab

Use the **Strand template** tab to define the vertical strand positions with grid points in a beam. Strands may only be located at the positions defined in this template.

Grid spacing along X

Use to define grid spacing values for the x axis. Strands are defined by their row and column number within the grid.

Separate values with commas or spaces. The first value is the distance from the left edge of the spandrel. Each successive value is the distance to the next grid point.

Grid spacing along Y

Use to define grid spacing values for the y axis. Strands are defined by their row and column number within the grid.

Separate values with commas or spaces. The first value is the distance from the the bottom edge of the spandrel. Each successive value is the distance to the next grid point.

NOTE Use multiplier to define a repeating series of values. For example, 8*2 indicates eight holes with a two unit spacing.

Strand pattern tab

Use the **Strand pattern** tab to define the placement of the strands at the grid points.

External file

With an external CSV file you can define an unlimited amount of individual strands.

Select to use an external file from the **Use external file** list.

Select an external CSV file by clicking **...** button.

Rebar end is same as start

Define the strand input.

- Select **Yes** to use the row and debond information of the strand start for the strands at the strand end.
- Select **No** to define an alternate row and debond information at the end of each strand.

Strand pattern definitions

Define the row and column for each end of the strand using the strand template positions in the strand grid. Strands may have different grid row positions at each end, but a strand cannot cross grid columns.

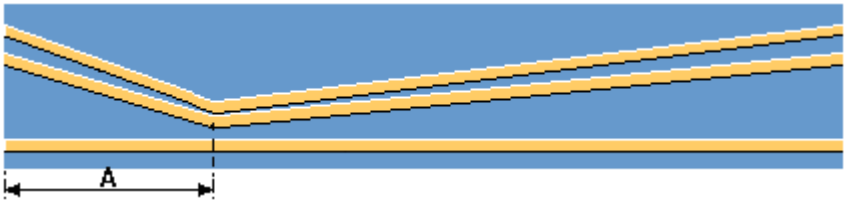
Option	Description
Strand start - Row	Define the horizontal row number in the strand template for the start of the strand.
Strand start - Debond	Define the debond length at the start of the strand.
Strand end - Row	If **Same as left** is **No**, define the horizontal row number in the strand template for the end of the strand.
Strand end - Debond	If **Same as left** is **No**, define the debond length at the end of the strand.
Middle debond - To start	Define the debond length from middle to the start of the strand.
Middle debond - To end	Define the debond length from middle to the end of the strand.
Columns	Define a list of one or more column numbers and/or column ranges in the strand template at which a strand should be created.
Size	Define the size of the strands by selecting if from the reinforcing bar catalog.
Grade	Define the strand grade.
Pull	Define the pull force of the strands.

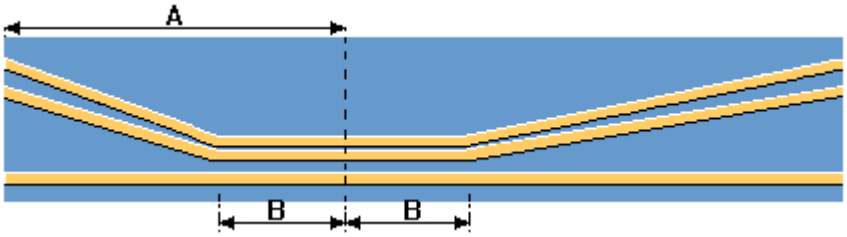
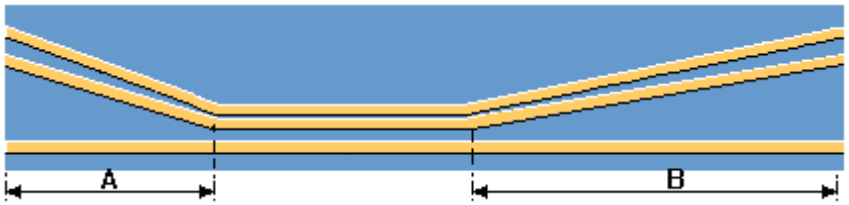
Strand profile tab

Use the **Strand profile** tab to define the profile for strands.

Strand depression

Strands can be straight or depressed. Use one or two control points to depress the strands. All strands use the same general profile.

Option	Description
	Straight
	Depressed in 1 point

Option	Description
	Depressed in 2 points
	Depressed in 2 fixed points

Depress position

Define the depress positions A and B either as a fixed distance or as a percentage of the member length.

Strand ht @ position

Define the height of the strand.

Attributes tab

Use the **Attributes** tab to define naming and numbering of the strands.

Option	Description
Prefix	Prefix for the part position number.
Start number	Start number for the part position number.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Strand code

Store strand code – With option **Store strand code** you may choose to store (or not) a user-defined textual attribute `TS_STRAND_CODE` to part.

Strand code – In field **Strand code** you may enter a string template for the value of the above mentioned user-defined attribute. The default template is the total number of created strands. The template may contain any text and the predefined fields for:

- %NUMBERS – is the numbers of created strands with different sizes and/or grades e.g. 10 (for one size & grade) or 6+4 (if two sizes and/or grades has been used).
- %SIZES – sizes of the strands e.g 1/2 or 1/2 + 3/8.
- %GRADES – grades of strands.

Strand creation at voids

Create strand at voids – The option **Yes** will show the strands even though the voids are present. The option **No** will not show the strand if void is present at any location along the strand path.

Select to cut the strands at void locations from **Cut strands at voids**.

Place strands perpendicularly

Select whether the strands are placed perpendicular to the beam start/end. This setting allows you to use the component on cross-modeled precast beams with different sections.

6.4 CS Components

Topics

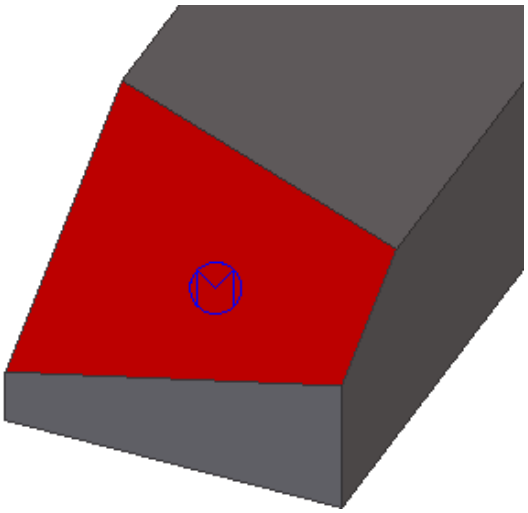
This chapter contains the following topics:

- 3D cut (10) (page 4208)
- Opening in wall (40) (page 4210)
- Rebar in beam (90) (page 4217)
- Border rebar for single edge (93) (page 4233)

3D cut (10)

The **3D cut (10)** component creates a 3D cut to a concrete part. The cutplane is determined by picking three points on the (concrete) part edges. The cut direction can be set. Also an cut offset can be set.

Use for

Situation	More information
	This component creates a cut through three points. The side to be removed can be defined.

Before you start

Before applying the macro, create the following part:

- Concrete part (panel, column, strip footing)

Defining properties

The component contains only one tab, named **Picture**.

Tab	Content
Picture	Defining the cut direction. Possibly set an cutplane offset.

Picking order

1. Concrete part.
2. Three points to determine the cut plane.

Picture tab

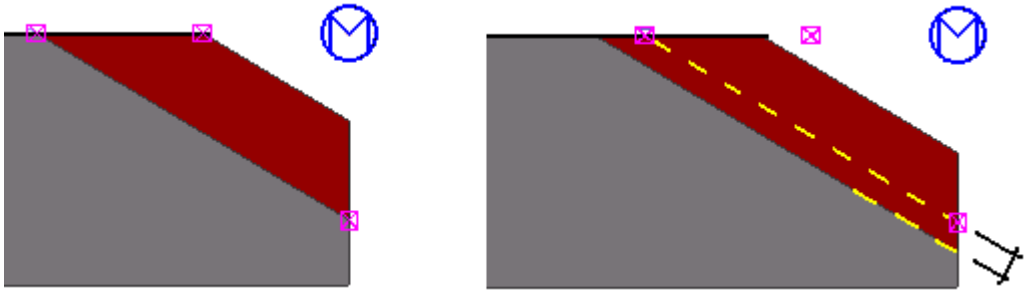
You can define the following settings on the **Picture** tab.

Cut type

Option	Description
Cut right	The parts' right side from the cut plane will be removed.
Cut left	The parts' left side from the cut plane will be removed.
Fit plane	The removed part is the shortest part (from fit plane to part's end).
No cut	No cut is applied.

Offset

You can define the offset for the cut plane. The offset distance is always perpendicular to the cut plane.



Explode component

You can select to explode the component if you do not need to modify the component after applying it. If you plan to modify the component later, select not to explode it.

Opening in wall (40)

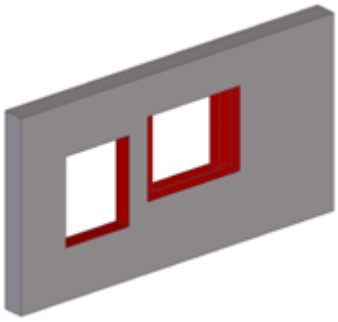
Opening in wall (40) creates an opening in a concrete wall of adjustable size, shape and position. Can also add an insulation panel to the wall.

Objects created

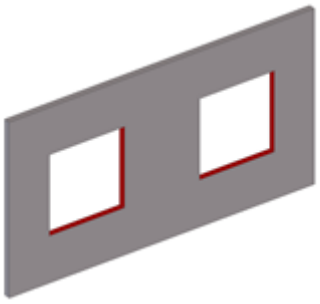
- Opening
- Insulation (optional).

Use for

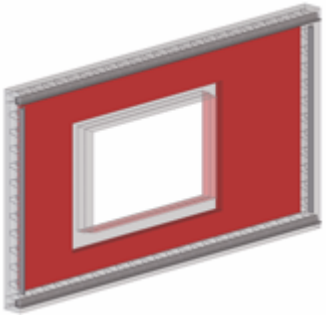
Square or circular opening in a wall and adding insulation to the wall.



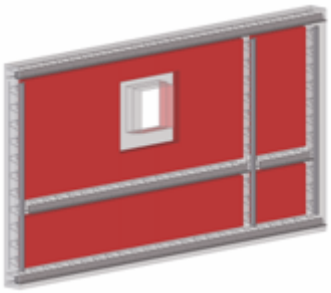
define rabbet



several openings



adding insulation



braced girders

TIP You can use Hole Generation (32) (page 3696) to create an opening with an irregular shape.

Before you start

Create the wall for example as a concrete panel or column.

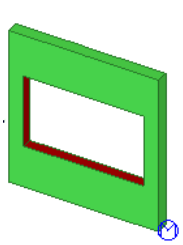
Opening tab

Use the **Opening** tab to define the shape and size of the opening.

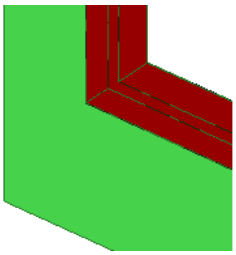
Type

First you can define the type of the opening. The options are:

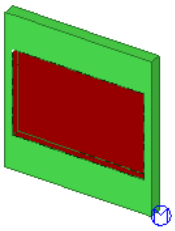
- Opening
- Opening no rabbet
- Opening with rabbet
- Opening only outside
- Opening only inside



Opening



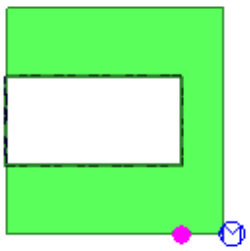
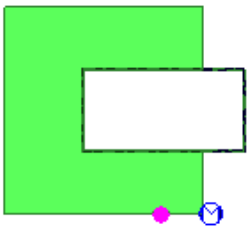
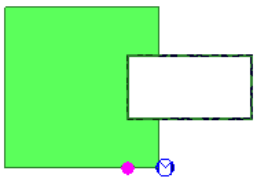
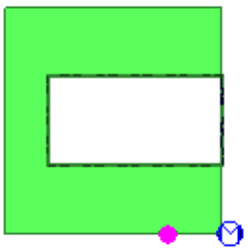
Opening with rabbet

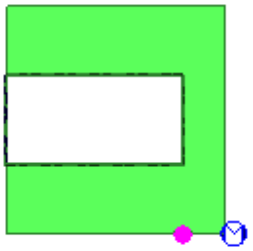
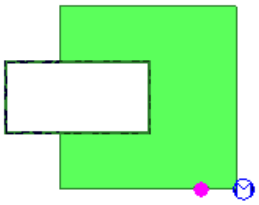
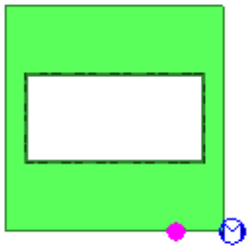
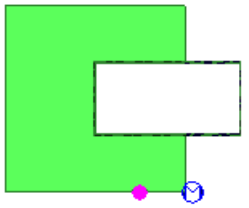


Opening only outside

Horizontal position

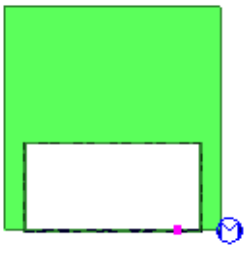
The horizontal reference point for creating the opening can be selected from a picklist. In the table below are the available options.

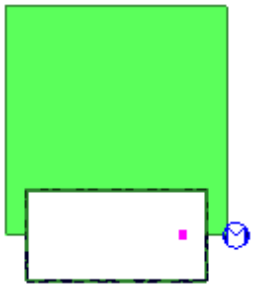
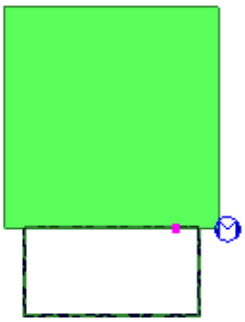
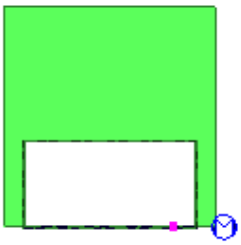
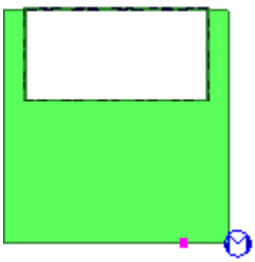
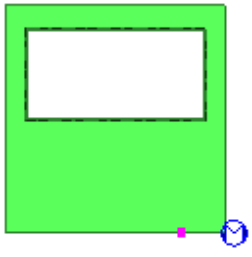
Option	Description	Example
Point is left side opening	The opening is created on the left side from the component insertion point.	
Point is middle side opening	The middle of the opening is created from the component insertion point.	
Point is right side opening	The opening is created on the right side from the component insertion point.	
Start panel	The opening is created from the startpoint of the wall.	

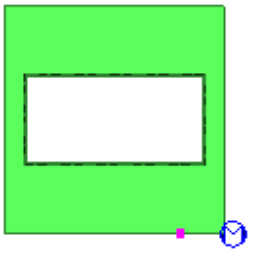
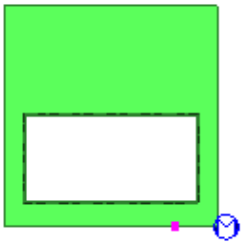
Option	Description	Example
End panel	The opening is created from the endpoint of the wall.	
Middle panel = left opening	The center of the wall is referencepoint. The opening is created to the left side.	
Middle panel = middle opening	The center of the wall is referencepoint. The opening is also created from the middle.	
Middle panel = right opening	The center of the wall is referencepoint. The opening is created to the right side.	

Vertical position

The horizontal reference point for creating the opening can be selected from a picklist. In the table below are the available options.

Option	Description	Example
Pos point bottom opening	The opening is created on the left side from the component insertion point.	

Option	Description	Example
Point is middle opening	The middle of the opening is created from the component insertion point.	
Pos point top opening	The opening is created on the right side from the component insertion point.	
Bottom panel	The opening is created from the startpoint of the wall.	
Top panel	The opening is created from the endpoint of the wall.	
Middle panel = bottom opening	The center of the wall is referencepoint. The opening is created to the left side.	

Option	Description	Example
Middle panel = middle opening	The center of the wall is referencepoint. The opening is also created from the middle.	
Middle panel = top opening	The center of the wall is referencepoint. The opening is created to the right side.	

Horizontal/Vertical offset

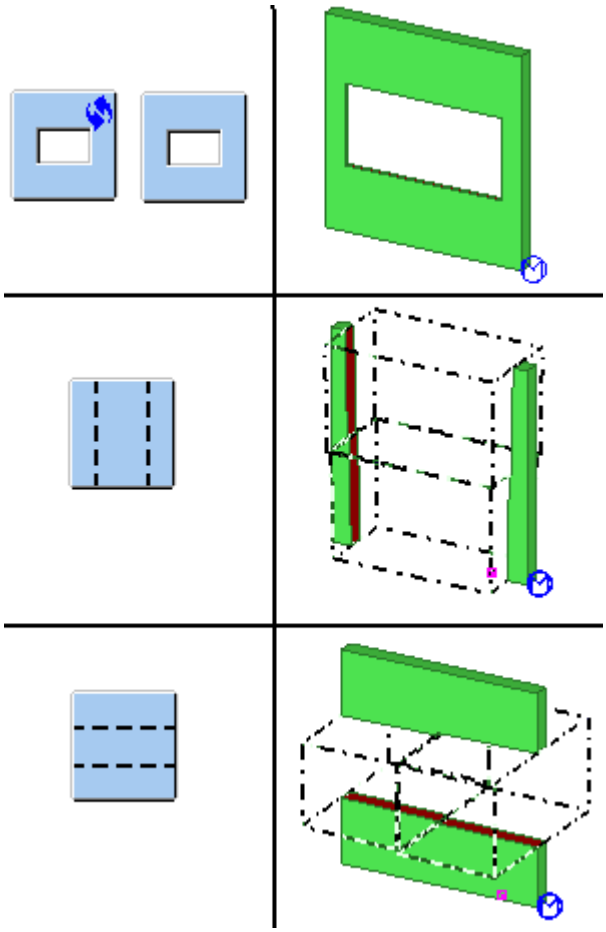
The opening can be moved vertically and horizontally by using these two offset value fields.

Rabbet - side

With this picture picklist you can define the side where the rabbet is created.

Cut - rotation

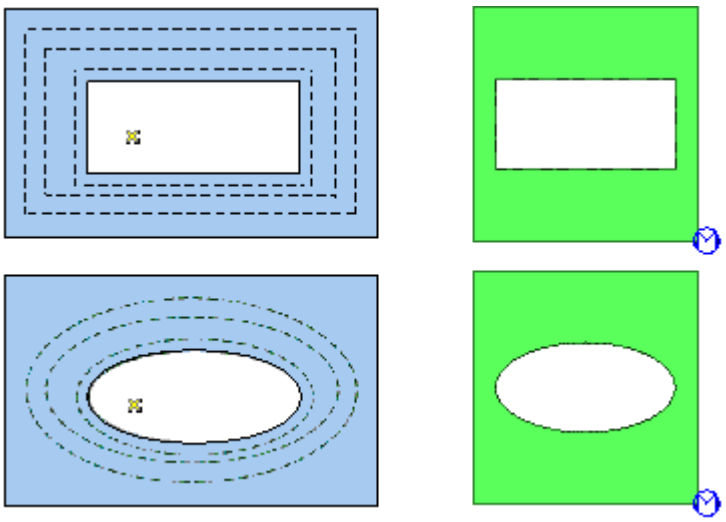
With this picture-picklist you can define the rotation of the cut.



Shape of the opening

With the picklist you can choose the general shape of the opening in the wall.

The default shape is a rectangular opening. The other shape is the circular shape.



Insulation tab

Use the **Insulation** tab to define insulation thickness, offset and creation of partcut.

The option 'Cut opening size' can be used to define if the insulation should be cut at the opening in the wall.

Option	Description
t b h	Define the thickness, width, and height of the part.
Pos_No	Define a prefix and a starting number for the part position number. Enter the assembly position number on the second row. The default part start number is defined in the **Components** settings in **File menu** --> **Settings** --> **Options** .
Material	Define the material grade. The default material is defined in the **Part material** box in the **Components** settings in **File menu** --> **Settings** --> **Options** .
Name	Define a name that is shown in drawings and reports.
Class	Enter a number to group the parts that the component creates. By default, the class number affects the color in which the part is shown in model views.

Parameters tab

Use the **Parameters** tab to create assembly panel insulation.

Possiblilty to weld the insulation to the panel to get an assembly.

Three options available: **Default**, **Yes** and **No**. **Default** option is same as **Yes**.

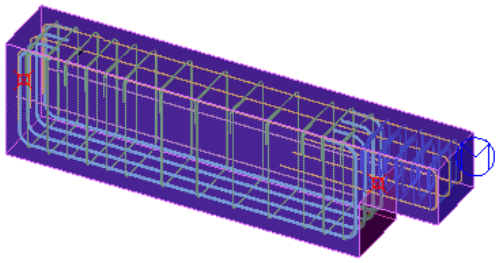
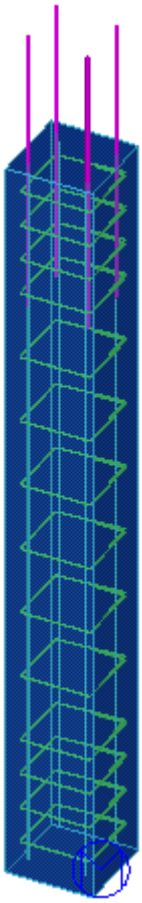
Rebar in beam (90)

Rebar in beam (90) adds reinforcement bars to concrete beams.

Parts created

Complete reinforcement structure: stirrups and perpendicular rebars are created in the concrete part.

Use for

Situation	More information
	Rebars in concrete beam.
	Rebars in a concrete column.

Do not use for

Use this component to add reinforcement bars to a concrete part.

Before you start

Before the component can be created, you need to create a concrete part first.

Primary bottom bars tab

Use the **Primary bottom bars** tab to define the properties of the lower reinforcing bars.

Properties

Option	Description
Grade	Defines the grade of the main bottom reinforcing bars. This field cooperates with the **Size** field.
Size	Defines the diameter of the main bottom reinforcing bars. Pressing the **...** button right of the field will open the **Select rebar** dialog box. In the dialog box you can select the grade and the accompanying diameter. **NOTE** Selecting a size will override the value in the **Grade** field.
Bend lengths left	Defines the length of the left end-extension.
Bend lengths right	Defines the length of the right end-extension.

End conditions left/right

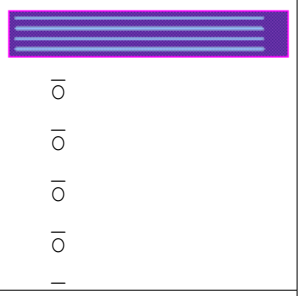
Picklist to define the rebars' end condition on the left and right side. The options are:

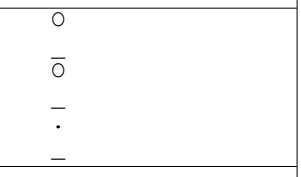
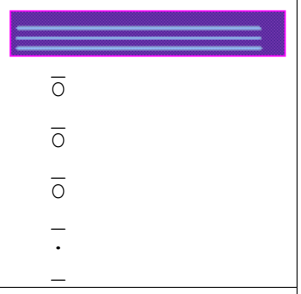
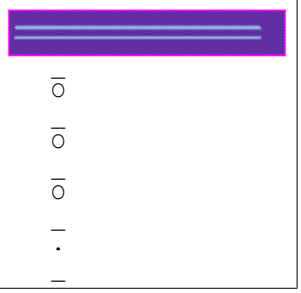
Picklist item	Example
Default	
90 Degree	
135 Degree	

Picklist item	Example
180	
-180	
-135	
-90	
-45	
45	

Bar not to create

Picklist to define which rebar should not be created. The options are:

Picklist Item	Example
	

Picklist Item	Example
	
	
	

Creation method

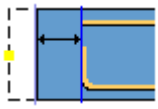
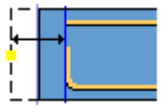
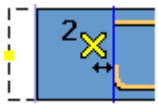
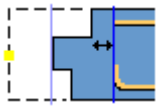
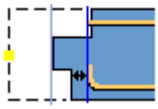
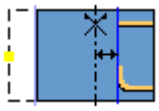
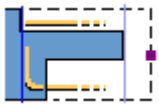
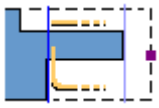
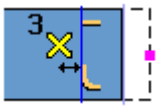
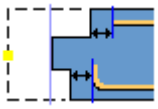
This field defines the way the rebars are created. There are three options:

Option	Description
No rebars	No bottom rebars are created.
Number of bars	A fixed numer of bars is created. The spacing between the bars is automatically calculated.
By spacing	A accompanying field becomes active. The entered value is the fixed spacing for the rebars. The number of rebars is calculated automatically.

Positioning and distances

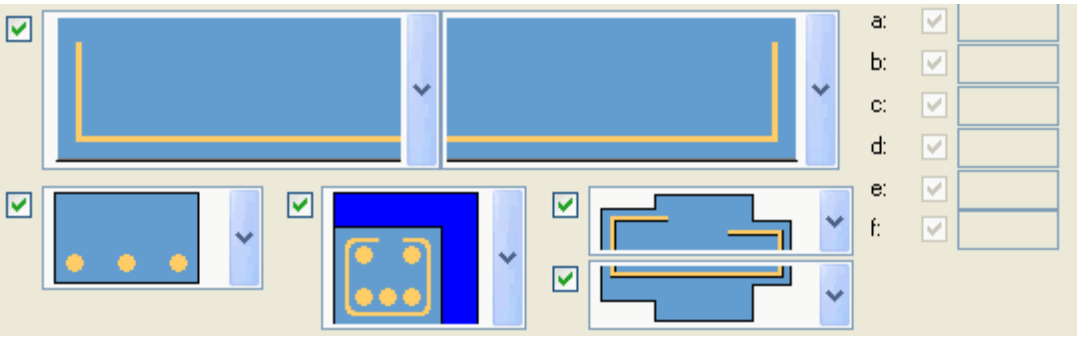
In the right upper side of this tab a picture and some picklists are displayed. These fields and lists are to define the position and de rebar distances.

The distance fields are used for defining the size of the rebars. The picklists are for positioning. Below some examples for the positioning rebars on the left side:

Picklist item	Description	Example
	The part edge is normative for positioning the rebar.	
	The concrete parts' reference point is normative for positioning the rebar.	
	The picked position is normative.	
	The edge of the top part cut is normative for positioning the rebar.	
	The edge of the bottom part cut is normative for positioning the rebar.	
	The centerline of the concrete part is normative for positioning the rebar.	
		
		
		
		

Shape and pattern

The picklists and input fields are used to define the rebar shape, rebar pattern and rebar distribution.



In the left bottom side of the tab, some picklists are available for defining the shape and distribution of the bottom rebars.

A - Shape of rebar

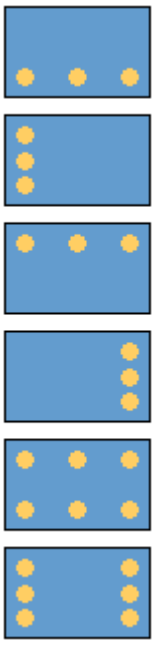
These picklists define the common shape of the rebars on both ends of the concrete part. For some shapes distances can be entered (B).

B - Distance fields

These distance fields only apply for two rebar shapes (A).

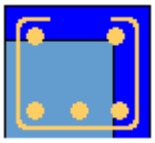
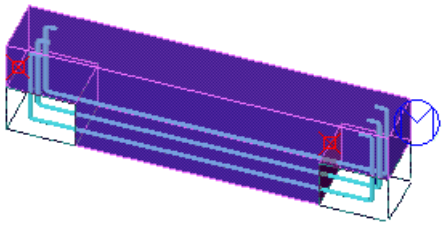
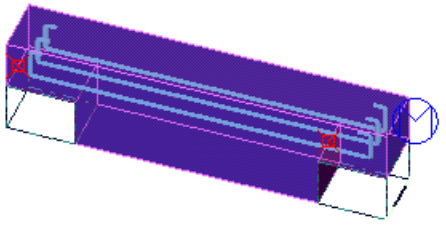
C - Rebar pattern

Picklist for defining the rebar pattern. The picklist contains six options:



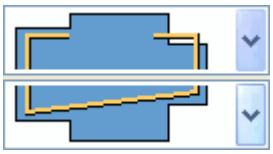
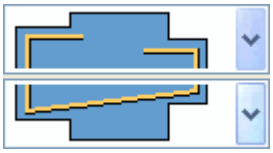
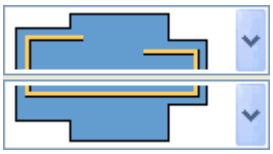
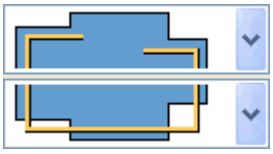
D - Consider part cuts in longitudinal direction

This picklist defines if part cuts in the concrete part should be taken into account when positioning the rebars.

Picklist item	Example
	
	

E - Consider part cuts in vertical direction

These picklists contain options for defining if rebar shape should consider the partcuts in vertical direction. For both top side and bottom side the placing can be defined. Examples:

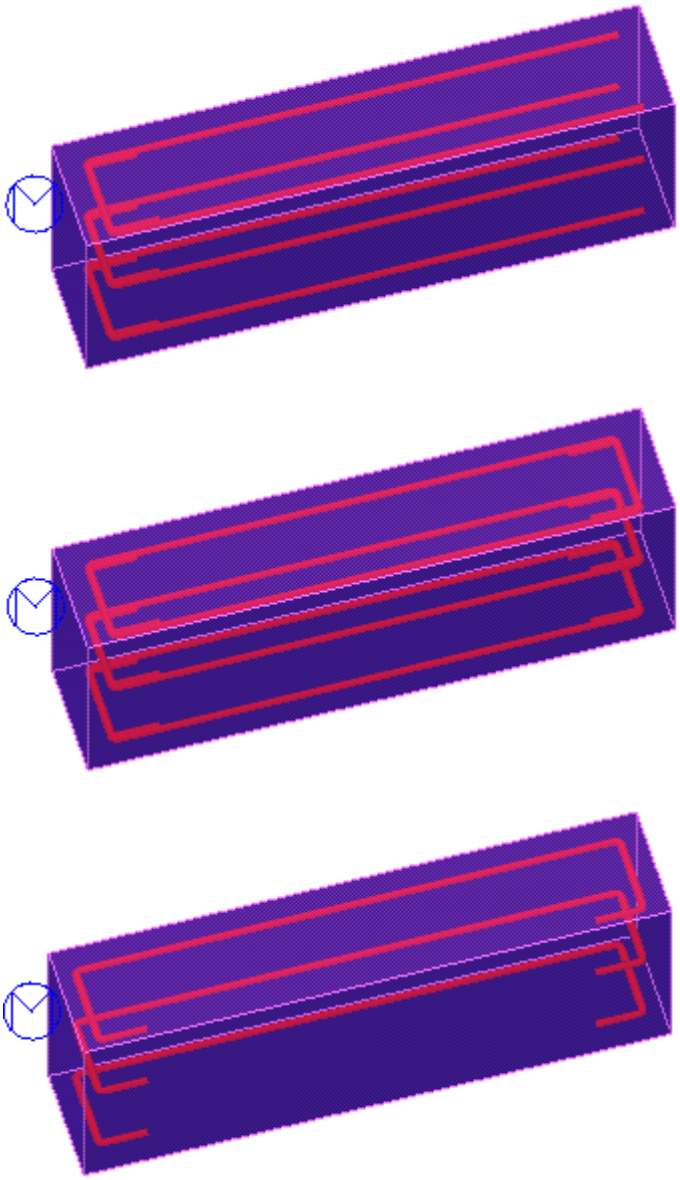
Picklist item	Example
	
	
	
	

Side left tab

Use the **Side left** tab to create and define the properties of the horizontal rebars on the left side.

The rebars which can be created are placed in longitudinal direction (just as the Primary bars), but now the bends are created horizontally. The options on this tab (defining rebar shape, dimensions, number of bars, and rebar distribution) are similar to the options on the **Primary bottom bars** tab.

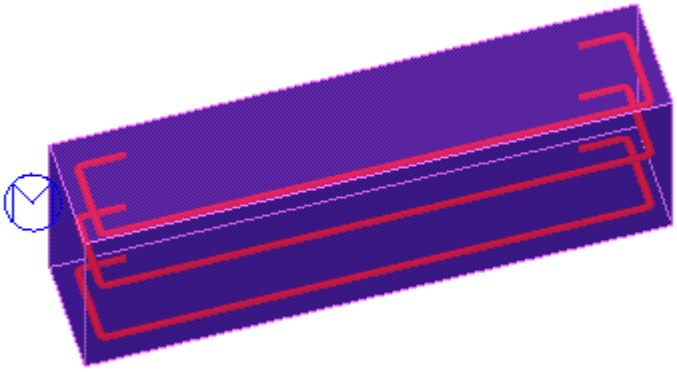
Examples of rebars which can be created with this tab:



Side right tab

Use the **Side right** tab to create and define the properties of the horizontal rebars on the right side.

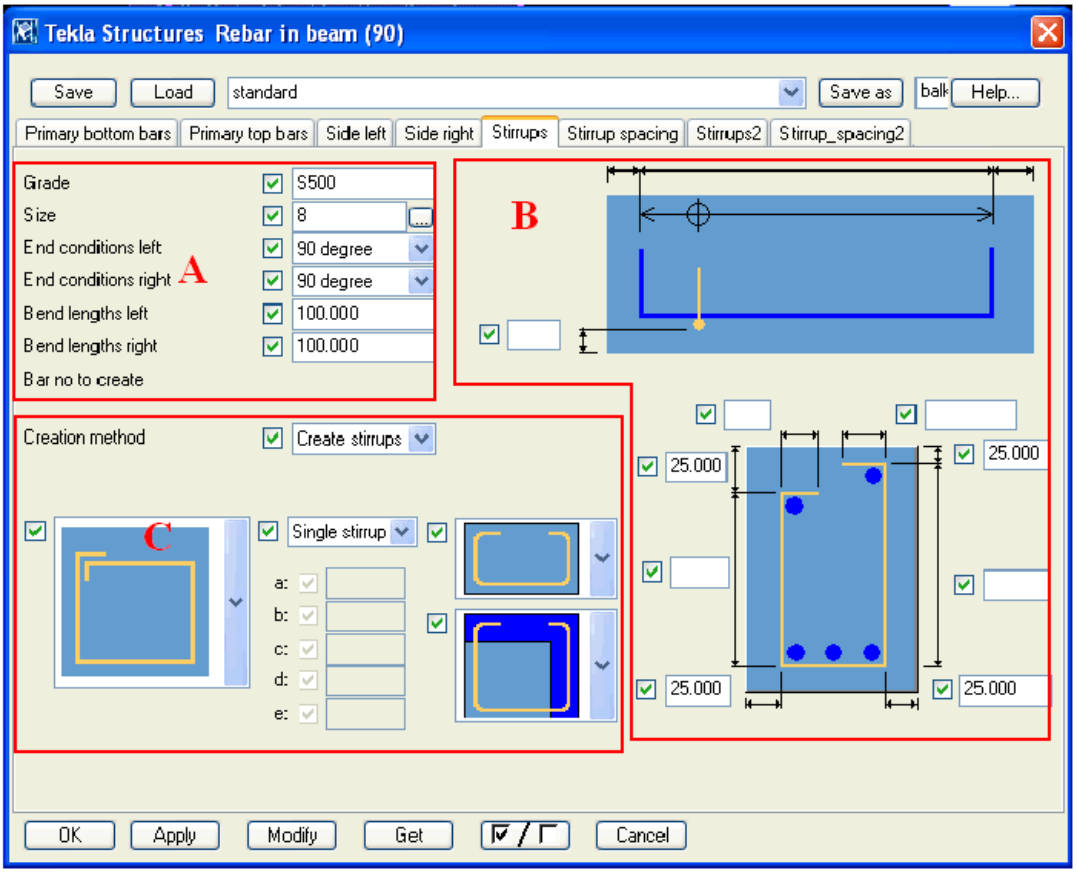
Similar to the **Side left** tab, but now the reference is the parts' right side.



Stirrups tab

Use the **Stirrups** tab to define the dimensions of the stirrups in first section of the concrete part.

In the picture below, the options are grouped.



Section A - Stirrup dimensions

Option	Description
Grade	Defines the grade of the main bottom reinforcing bars. This field cooperates with the **Size** field.
Size	Defines the diameter of the main bottom reinforcing bars. Pressing the **...** button right of the field will open the **Select rebar** dialog box. In the dialog box you can select the grade and the accompanying diameter. **NOTE** Selecting a size will override the value in the **Grade** field.
Bend lengths left	Defines the length of the left end-extension.
Bend lengths right	Defines the length of the right end-extension.

Section B - Stirrup dimensions

One input field is available for defining a fixed distance from concrete parts' bottom side to bottom side of stirrup.

NOTE This offset is normative.

The input fields in the bottom side of this section are for defining the offset, width and height of the stirrups.

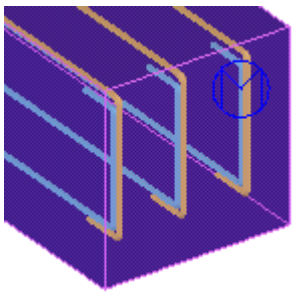
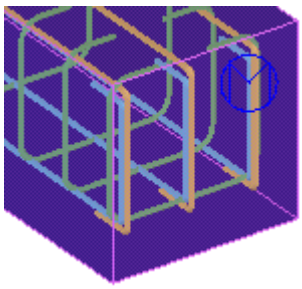
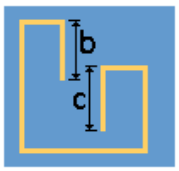
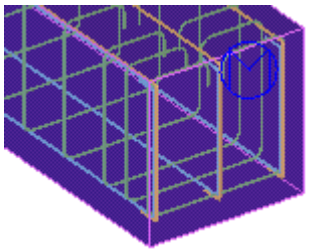
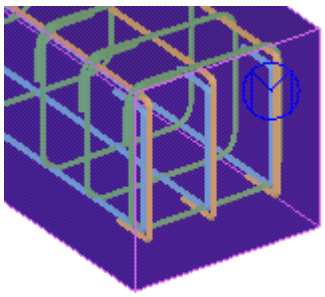
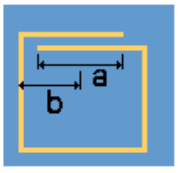
Section C - Stirrup shape

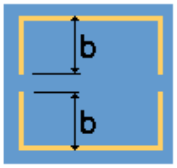
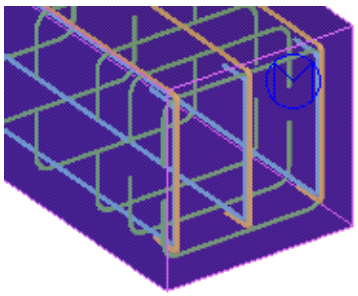
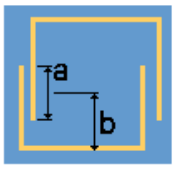
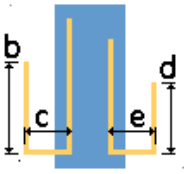
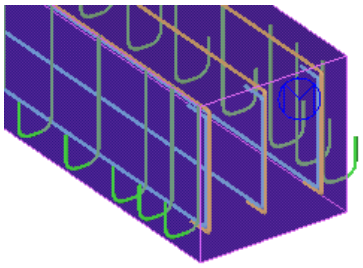
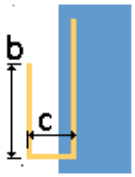
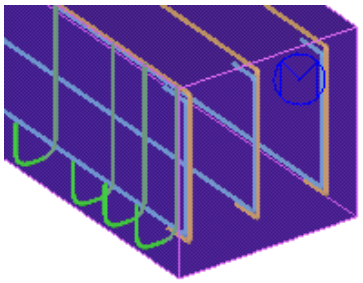
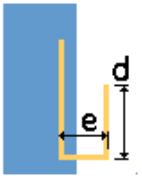
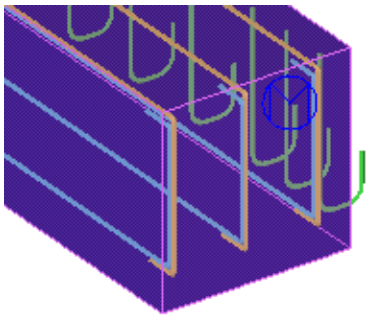
In this section the general shape of the stirrups is defined.

First there is the **Create stirrups** option. This picklist has three options:

- **No** - No stirrups are created
- **As one group** - Stirrups are created as one group
- **As separate group** - Stirrups are created as separate groups

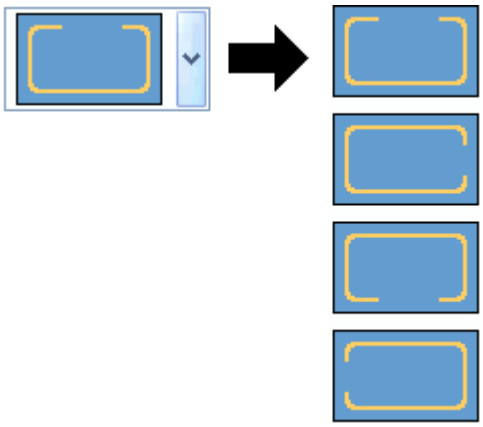
The general shape of the stirrup can be defined. You can choose a shape from the picklist:

Picklist item	Description	Example
	No stirrups are created	
	U-shape stirrups	
	C-shape inside	
	Closed box	
	Closed box overlap	

Picklist item	Description	Example
	Divided stirrup	
	Divided stirrups overlap	
	Double U-shape	
	Single U-shape left	
	Single U-shape right	

Stirrup rotation

The rotation of the stirrups can be set with below picklist. The picklist contains four options: each option will rotate the stirrup 90 degrees counterclockwise.



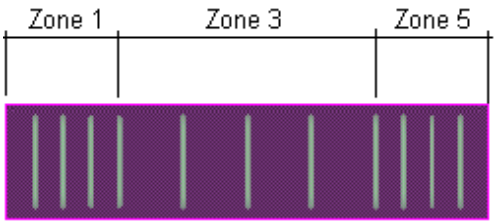
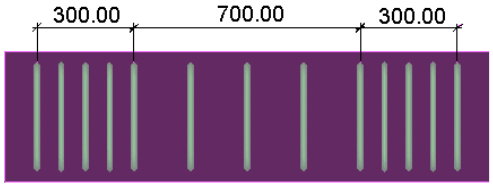
Stirrup spacing tab

Use the **Stirrup spacing** tab to define the distribution of the stirrups along the concrete part.

You can define six zones for the distribution. For each zone you can set a number of stirrups. The distance between the stirrups can be defined in two ways:

- Fixed spacing between each stirrup.
- Distance between outer stirrups. (Spacing is then calculated from this distance divided by the number of stirrups.)

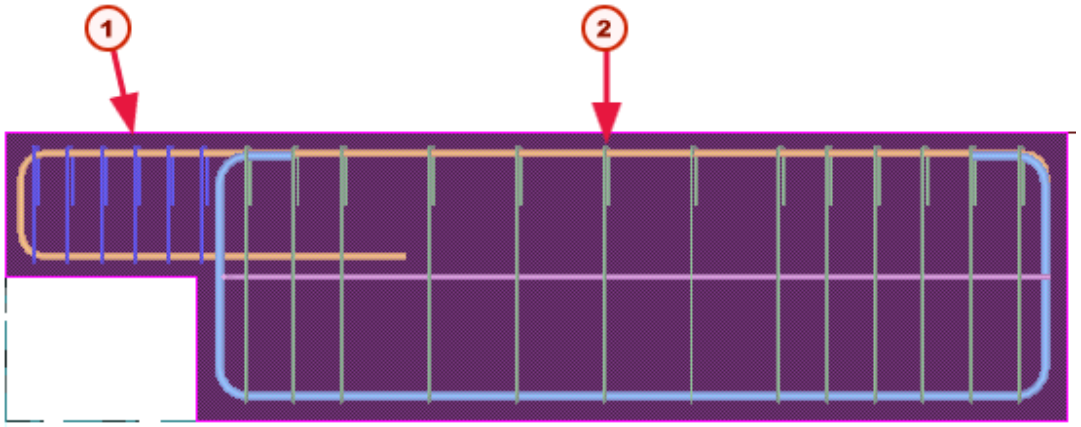
Examples:

Picklist item	Example
Main stirrups Number of stirrups <space> Length Zone 1 3 100.000 Zone 2 Zone 3 4 Zone 4 Zone 5 3 100.000 Zone 6	
Main stirrups Number of stirrups <space> Length Zone 1 4 300.000 Zone 2 Zone 3 4 700.000 Zone 4 Zone 5 4 300.000 Zone 6	

Stirrups 2 tab

Use the **Stirrups 2** tab to define a second group of stirrups.

This tab can be used if the shape of the concrete part is not uniform. For example, if a part cut is applied, you may need different stirrups on that area.



1 Stirrups 2

2 Stirrups

Stirrups 2 spacing tab

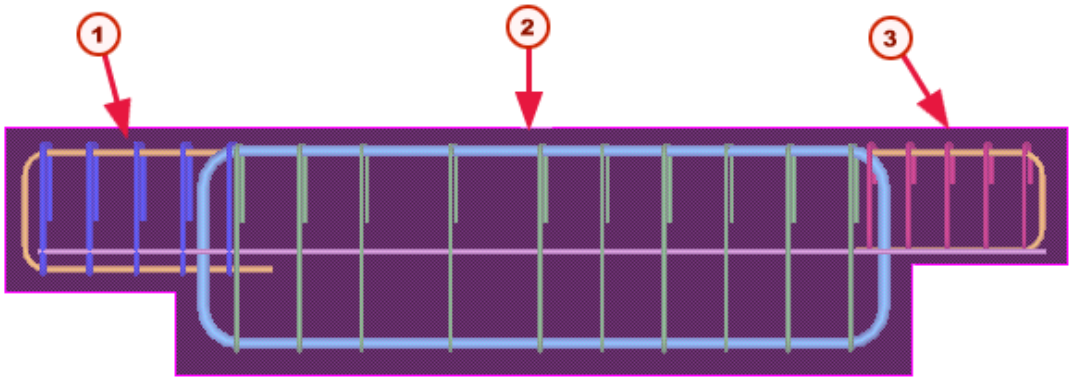
Use the **Stirrups 2 spacing** tab to define the spacing of the second group of stirrups.

Stirrups 3 tab

Use the **Stirrups 3** tab to define a third group of stirrups.

You may need this tab in case the concrete part contains multiple part cuts. For each area, the properties of the stirrups can be configured.

Example:



Stirrups 2

Stirrups

Stirrups 3

Stirrups 3 spacing tab

Use the **Stirrups 3 spacing** tab to define the spacing of the third group of stirrups.

Advanced tab

Use the **Advanced** tab to define the naming and numbering properties of rebars and stirrups.

The following information can be added:

- **Comment**
- **Name**

Rebar profile	Default Name
Main bottom rebars	RB
Main top rebars	RB
Side left	RB
Side right	RB
Stirrups	STIRRUP
Stirrups2	STIRRUP
Stirrups3	STIRRUP

- **Class**

  If the **Class** fields are left blank, then the following default classes will be used:

Rebar profile	Default Class
Main bottom rebars	201
Main top bars	202
Side left	203
Side right	204
Stirrups	301
Stirrups2	302
Stirrups3	303

- **Series**

  If this field is left blank, the prefix will be empty for all rebar profiles.

- **Start number**

  If this field is left blank, Tekla Structures will use '1' as default start number for all rebar profiles.

Configuration tab

Use the **Configuration** tab to define the bending radius and rotation.

Bending radius

The bending radius can be set separately for the main reinforcing bars and the stirrups. Furthermore the radius can be set separately for the main bends and the hook-bends.

The bending radius can be determined in three ways:

Picklist item	Description
rebar_database.inp	The bending radius is determined from a configuration file `rebar_database.inp`. This file is located in the `profil` folder.
Relative to diameter	For this option the radius is calculated by multiplying the reinforcing bar diameter with the entered coefficient.
Bending radius	Use this option to use a fixed value for the bending radius.

Rotation

The picklist determines the orientation of the complete rebar structure in the concrete part. The picklist contains four options, each item representents a 90 degree rotation.

Border rebar for single edge (93)

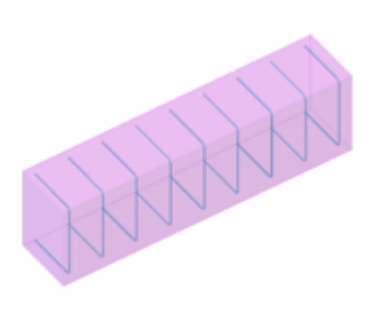
Border rebar for single edge (93) creates hairpin rebars in a concrete panel or wall.

Parts created

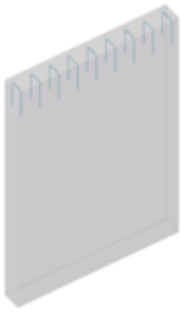
Hairpin rebars.

Use for

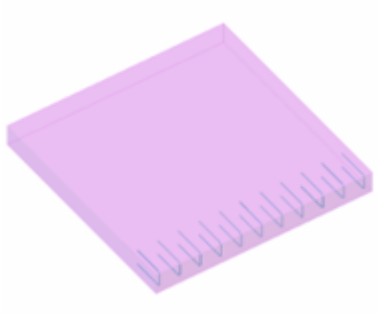
Use this component for creating hairpin rebars in a concrete panel.



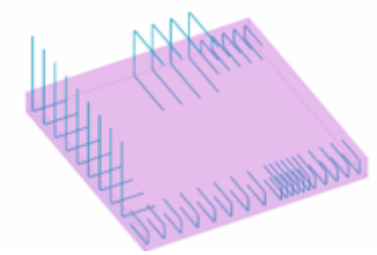
default beam



default panel



default floor



border reinforcing bars

Before you start

Before applying the component, the following parts must be created:

- Concrete panel or wall

Pins tab

Use the **Pins** tab to define the properties of the lower reinforcing bars.

Properties

Option	Description
Grade	Defines the grade of the main bottom reinforcing bars. This field cooperates with the **Size** field.
Size	Defines the diameter of the main bottom reinforcing bars. Pressing

Option	Description
	the **...** button right of the field will open the **Select rebar** dialog box. In the dialog box you can select the grade and the accompanying diameter. **NOTE** Selecting a size will override the value in the **Grade** field.
Bend lengths left	Defines the length of the left end-extension.
Bend lengths right	Defines the length of the right end-extension.

End conditions left/right

Picklist to define the rebars' end condition on the left and right side. The options are:

Picklist item	Example
Default	
90 Degree	
135 Degree	
180	
-180	

Picklist item	Example
-135	
-90	
-45	
45	

Pin spacing tab

Use the **Pin spacing** tab to define the distribution of the hairpins along the concrete part.

You can define six zones for the distribution. For each zone you can set a number of hairpins. The distance between the hairpins can be defined in two ways:

- Fixed spacing between each stirrup.
- Distance between outer hairpins. (Spacing is then calculated from this distance divided by the number of hairpins).

Advanced tab

Use the **Advanced** tab to define additional information to the hairpins.

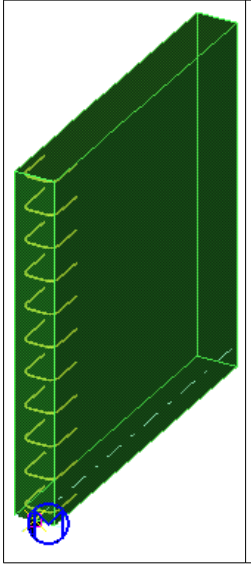
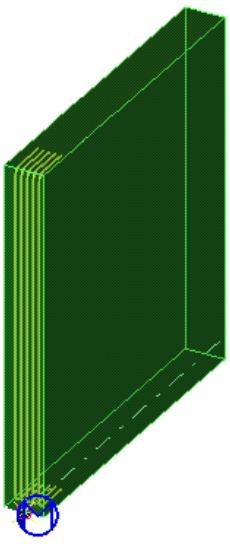
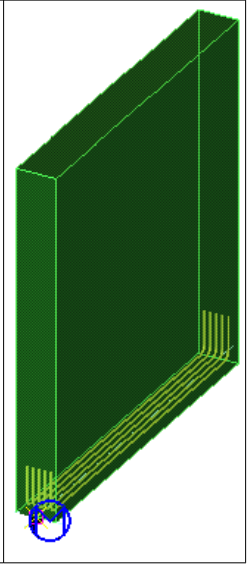
Option	Description
Comment	Add a comment about the part.
Name	Name that is shown in drawings and reports.
Class	Part class number.
Serie	Prefix for the part position number.
Start number	Start number for the part position number.

Configuration tab

Use the **Configuration** tab to define the bolt pattern and extra rotation.

Extra rotation

This picklist allows you to rotate the hairpin rebars. There are three options:

No	Around X	Around Y
		

Bending radius sec

The bending radius can be determined in three ways:

Picklist item	Description
rebar_database.inp	The bending radius is determined from a configuration file `rebar_database.inp`. This file is located in the `profil` folder.
Relative to diameter	For this option the radius is calculated by multiplying the reinforcing bar diameter with the entered coefficient.
Bending radius	Use this option to use a fixed value for the bending radius.

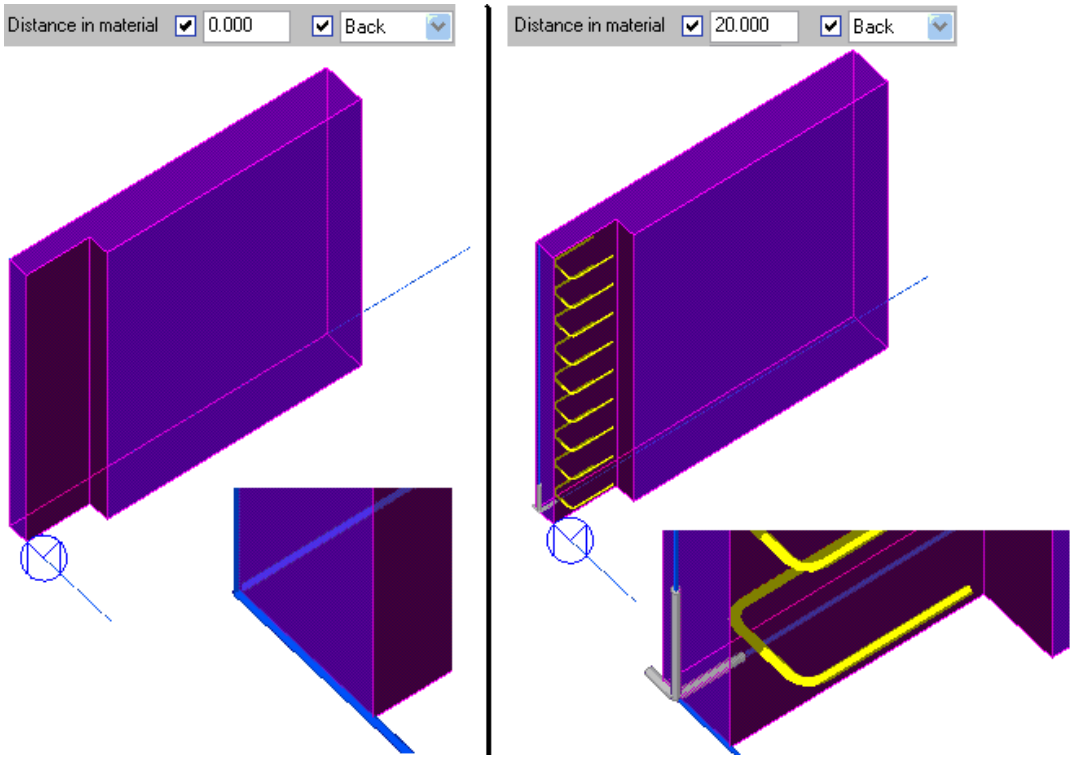
Detect outside geometry

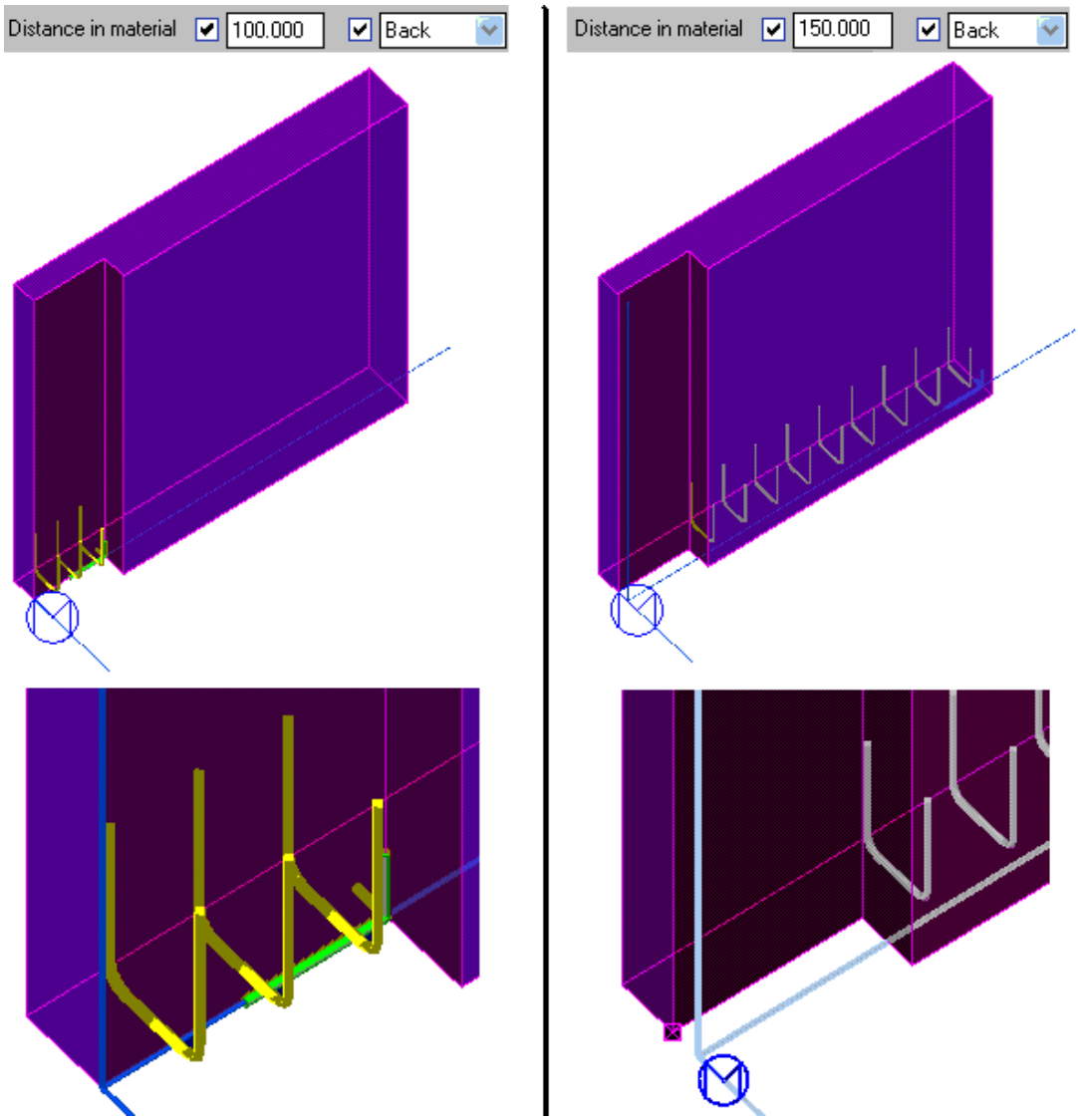
This parameter can be used to detect part cuts in the concrete part. The search area is defined with two parameters:

Distance in material and a **Position in plane** picklist.

Use these parameters to define the reference for the rebars. See the examples below for more clarification:

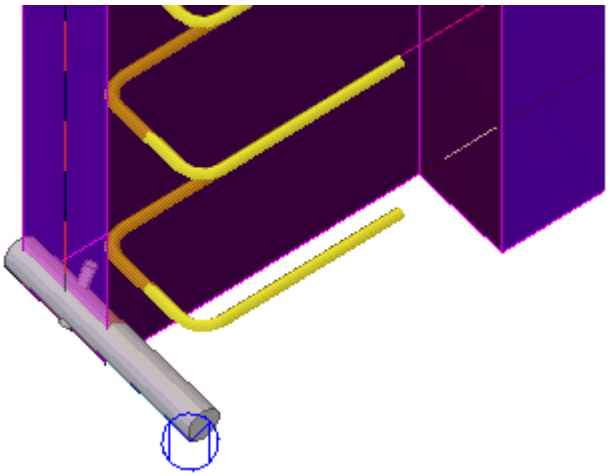
(The concrete panel has thickness 200 mm. The partcut depth is 100 mm.)





Draw geometry with profiles

Picklist to define if geometry profiles should be created or not. These geometry profiles define the rebar insertion point, which is based on the values for distance in material and can help setting up the rebar pattern.

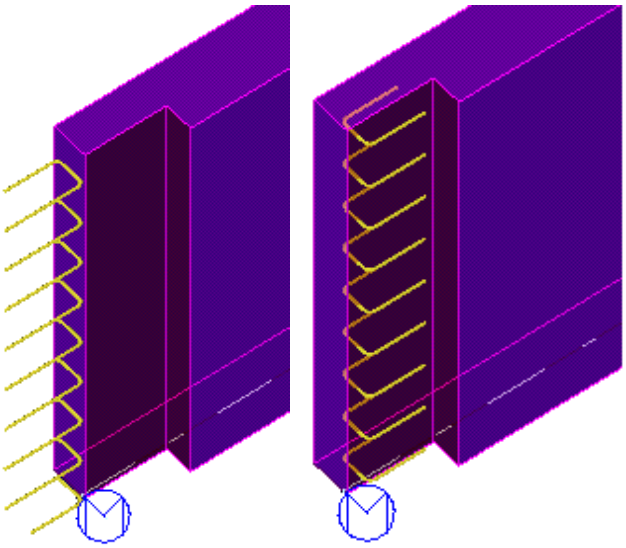


Detect negative volume

If a negative volume is applied to the concrete panel, you can define whether this negative volume should be recognised or not.

In below example a part cut is added with depth of 110 mm. If the value field is set to 0, the rebars can not be placed correctly.

If the value field is set equal or greater to the part cut depth (e.g. 110), the panel thickness will be recognised and the rebars are placed correctly.



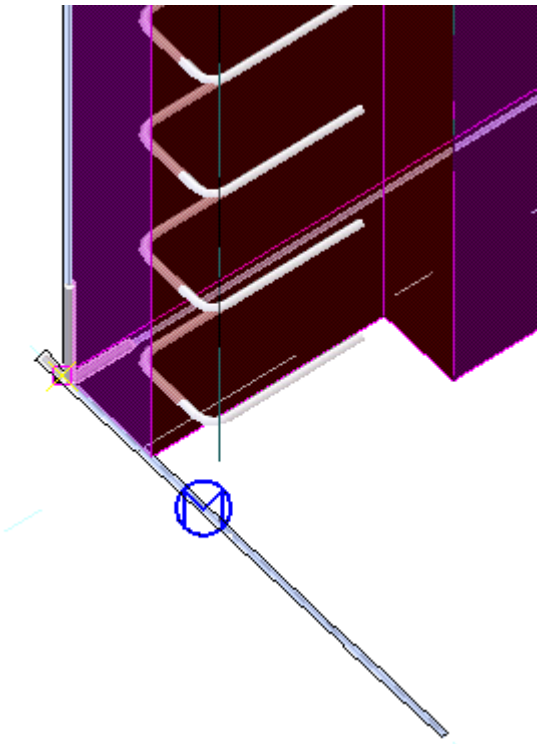
Difference in thickness = 0

Difference in thickness = 110

Draw axis

The **Draw axis** picklist has two options: **Yes** and **No**. If **Yes** is chosen, the component creates extra colored profiles. This can help to set up the component and gives more clearness in component orientation.

If **No** is chosen, the extra colored profile are omitted.



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